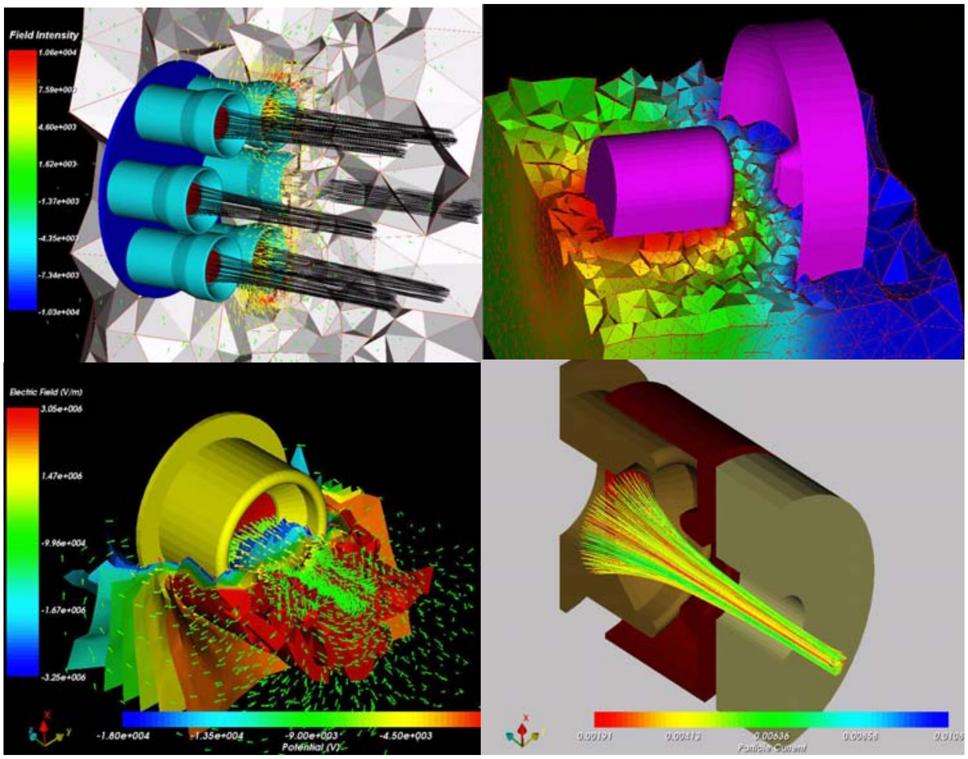
Beam Optics Analyzer, v4

User Manual Version 0.5
(May 2010 - updates in progress)


Field Intensity
1.06e+004
7.59e+003
4.60e+003
1.62e+003
-1.37e+003
-4.35e+003
-7.34e+003
-1.03e+004
Electric Field (V/m)
3.05e+006
1.47e+006
-9.96e+004
-1.67e+006
-3.25e+006
-1.80e+004
-1.35e+004
-9.00e+003
-4.50e+003
Potential (V)
0.00191
0.00413
0.00636
Particle Current

Calabazas Creek Research, Inc.
690 Port Drive, San Mateo, CA 94404

Table of Contents

1.0 Introduction 4

2.0 Installation 4

- 2.1 Minimum System Requirements 4
- 2.2 Software Requirements for Optimization Runs 4
- 2.3 Website Installation 5
- 2.4 Executing BOA GUI 5

3.0 Basic Operation 6

- 3.1 File Structure 6
- 3.2 Screen Organization 6
- 3.3 Project/Case Manager 8
  - 3.3.1 Project Menu 8
  - 3.3.2 Case Menu 9
- 3.4 File Menu 11
- 3.5 View Menu 12
- 3.6 Analysis Menu 18
- 3.7 Model Menu 19
- 3.8 Attributes Menu 20
  - 3.8.1 Electrostatic Attributes 21
  - 3.8.2 Beam Optics Attributes 23
  - 3.8.3 Magnetostatics Attributes 39
  - 3.8.4 Optimizer Attributes 45
  - 3.8.5 Misc Attributes 48
- 3.9 Meshing Menu 48
  - 3.9.1 Setting Mesh Parameters 48
  - 3.9.2 Generating and Displaying the Initial Mesh 55
- 3.10 Execute Menu 58
  - 3.10.1 Executing a Single Case 58
  - 3.10.2 Executing an Optimizer Case 61
  - 3.10.3 Monitoring a Beam Optics Run 62
- 3.11 View Results Menu 63
  - 3.11.1 Viewing Electrostatic Results 63
  - 3.11.2 Viewing Magnetostatic Results 73
  - 3.11.3 Viewing Beam Analysis Results 83
  - 3.11.4 Viewing Run Log 95
  - 3.11.5 Generating Case Summary 95
  - 3.11.6 Viewing Optimizer Log 95
- 3.12 Tools 96
  - 3.12.1 Creating an Injected Beam Profile (Cibo Tool) 96
  - 3.12.2 Measuring Distance 97
  - 3.12.3 Material Properties 98
  - 3.12.4 License Installer 99

4.0 Examples 100

- 4.1 Example 1 - JLab Klystron Gun 100

Table of Contents (Con’t)

4.1.1 Example 1A - Electrostatic Analysis 100
4.1.2 Example 1B - Electrostatic Analysis (with Adaptive Meshing) 116
4.1.3 Example 1C - Electrostatic Beam Analysis (with Thermionic Emitter) 123
4.2 Example 2 - Multi-stage Depressed Collector (MSDC) 140
4.2.1 Example 2A - Electrostatic Beam Analysis (with Injected Beam) 140
4.2.2 Example 2B - Electro/Magnetostatic Beam Analysis w/ Secondary Emissions154
4.3 Example 3 - Solenoid for a Multibeam Klystron (MBK) 163
4.3.1 Example 3A - Magnetostatic Analysis 163
5.0 Troubleshooting 175
6.0 Appendices 176
6.1 Appendix A - Input Files 177
6.1.1 Geometry File 177
6.1.2 Injected Emitter File 178
6.1.3 External Magnetic Field File 181
6.1.4 Excel Optimization File 182
6.2 Appendix B - Output Files 183
6.2.1 Planar Particle Output File (Papa output) 183
6.2.2 Terminal Particle Output File (Enpa output) 183
6.2.3 Cibo Output File 183
6.2.4 Case Summary File 183
6.3 Appendix C - Internal Files 183

Calabazas Creek Research, Inc.

(650) 312-9575

Fax: (650) 312-9536

Email: RLIves@CalCreek.com

www.CalabazasCreekResearch or www.calcreek.com

Copyright 2010

1.0 Introduction

Beam Optics Analyzer (BOA) is an advanced 3D charged particle simulation code with optimization. Its principle applications include the design of electron guns, collectors and charged particle devices. Features include:

- Incorporates finite element analysis with adaptive and user-controlled meshing
- Compatible with commercial 3D CAD programs
- User friendly, intuitive, graphical user interface (GUI)
- Integral electrostatic and magnetostatic solvers
- Import of spent beam and magnetics files
- Accurate simulation of grids
- Secondary electron emission
- Fully relativistic
- Includes space charge and self-magnetic fields
- Built-in materials database
- Optimization of beam optics design of guns, collectors and beam lines
- Optimization of electrostatic and magnetostatic circuits
- Optimization of multi-stage collectors
- Support for symmetry in model simulation
- Parallel processing

[TBD. Revise. Add more background, context, discussion of applications, discussion of what advantages BOA has compared to competitors; discussion/overview of the optimizer, etc.]

2.0 Installation

BOA is available via download from www.Calcreek.com. BOA is certified for operation in Windows 2000, Windows XP, Windows Vista and Windows 7.

2.1 Minimum System Requirements

- Operating System: Windows 2000, XP, Vista and Windows 7.
- Memory Requirements: 1 gigabyte of RAM or greater.

Boa supports multicore CPU Architecture.

2.2 Software Requirements for Optimization Runs

The following software programs are required for running Optimization cases:

- Microsoft Excel
- SolidWorks 3D CAD software

2.3 Website Installation

1. **Download BOA Beta Version** - To install BOA Beta Version, go to www.calcreek.com and click on *Products*. In the *Software Products* section, click on *Boa*. In the *Download BOA Beta Version and User Manual* section click on *Download BOA Beta Version.* You will have the option to *Open* or *Save* the installer. If you choose *Open*, the installer will install BOA to your specified location or the default directory, which is C:\Program Files\Beam Optics Analyzer. If you choose *Save*, you will be prompted for the directory in which to save the installer. Save the file to the desired directory, double-click it and then follow the on screen instruction to proceed with the installation.
2. **Obtain License File** - The BOA solver requires a license file. To obtain a 30-day demo license, please contact Thuc Bui at bui@calcreek.com with your name and address. When the license file BoaLicense.lic is received, place it in C:\Program Files\Beam Optics Analyzer\license. (You will be asked for the serial number from the disk drive containing the executable files. This number can be obtained by opening a DOS window on the drive where the executables are stored and typing ‘dir/p’. The serial number consists of eight alpha-numeric characters separated by a hyphen.)
3. **Download Latest BOA Patch** - To install the latest patch, if one is available, click on the *Download BOA Latest Patch* link to either *Open* or *Save* the patch. (The *Download BOA Latest Patch* link is located directly below the *Download BOA Beta Version* link on the webpage described in Step 1 above.) If you have chosen the default directory for BOA installation, the patch will self-extract to the proper location. If not, during the extraction you can browse to the bin folder under your BOA installation directory to unzip the patch.

2.4 Executing BOA GUI

If you chose to create a boaGui ‘shortcut’ during installation (on your desktop and/or in the startup menu), click on the boaGui icon (shown in Figure 1) to start the program.

FIGURE 1. BoaGui shortcut icon

Alternatively, doubleclick on the boaGui executable file in c:\Program Files\Beam Optics Analyzer. Figure 2 shows this file highlighted.

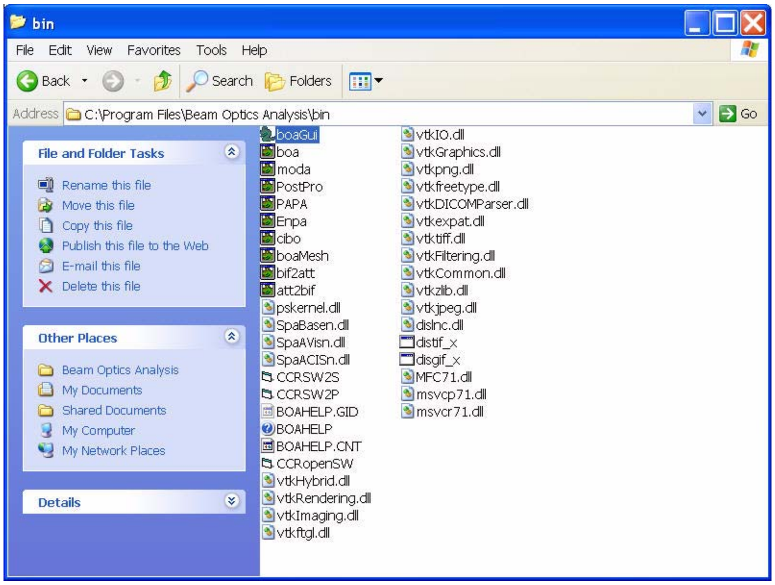

bin
File Edit View Favorites Tools Help
Back
Search
Folders
Address C:\Program Files\Beam Optics Analysis\bin
Go
File and Folder Tasks
Rename this file
Move this file
Copy this file
Publish this file to the Web
E-mail this file
Delete this file
Other Places
Beam Optics Analysis
My Documents
Shared Documents
My Computer
My Network Places
Details
boaGui
boa
moda
PostPro
PAPA
Enpa
cibo
boaMesh
bif2att
att2bif
pskernel.dll
SpaBasen.dll
SpaAVisn.dll
SpaACISn.dll
CCRSW2S
CCRSW2P
BOAHELP.GID
BOAHELP
BOAHELP.CNT
CCRopenSW
vtkHybrid.dll
vtkRendering.dll
vtkImaging.dll
vtkftgl.dll
vtkIO.dll
vtkGraphics.dll
vtkpng.dll
vtkfreetype.dll
vtkDICOMParser.dll
vtkexpat.dll
vtktiff.dll
vtkFiltering.dll
vtkCommon.dll
vtkzlib.dll
vtkjpeg.dll
dislnc.dll
distif_x
disgif_x
MFC71.dll
msvcp71.dll
msvcr71.dll

FIGURE 2. Contents of c:\Program Files\Beam Optics Analyzer directory. Doubleclick on boaGui to execute the Boa Gui.

3.0 Basic Operation

3.1 File Structure

Boa uses a file structure that includes both Cases and Projects. A Case consists of a single analysis and a Project is a logical grouping of Cases. Typically users group all the simulations (Cases) associated with a single device into one Project.

The Project directory contains the boa project file with the file extension *.boa. Case directories are located within the Project directory. Case directories contain geometry files (ACIS or a ParaSolid files), parameter files, BOA input files, and all output files. (See “Appendices” on page 176 for details on all input files, internal files and output files created by BOA.)

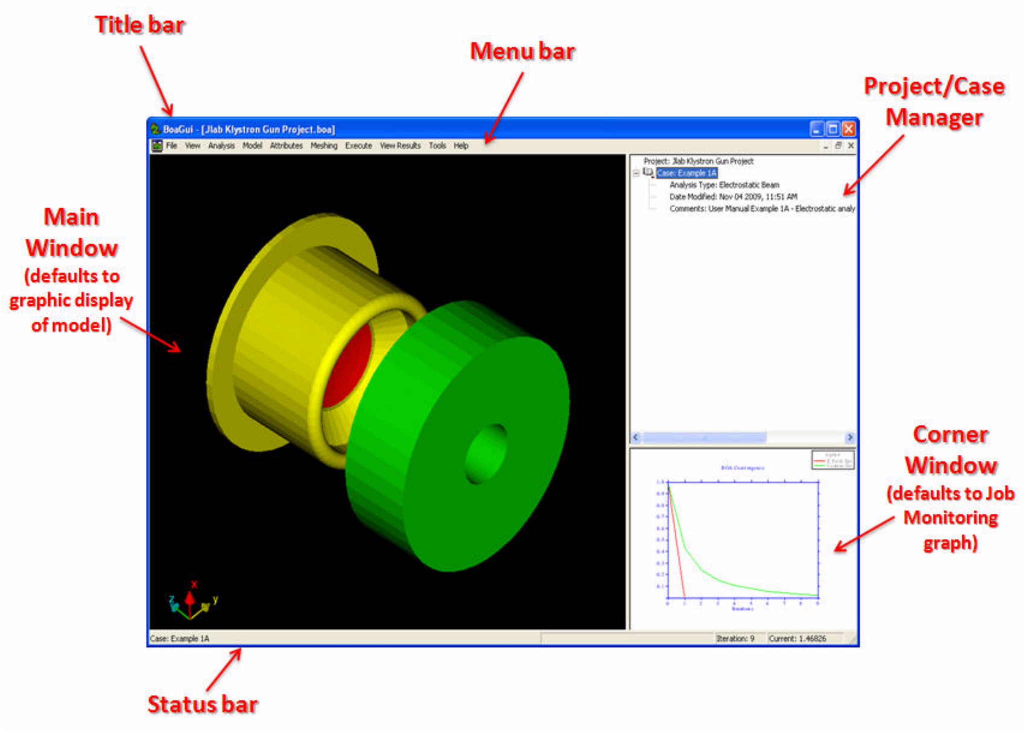
{TBD: maybe add a section discussing overview of all inputs & outputs? or is that unnecessary?}

3.2 Screen Organization

When you first execute BoaGui, the title bar and menu bar appear at the top of the screen and the section below the bars is blank. When you open a Project (described in the next section) the screen below the bars will divide into three section as shown in Figure 3:

- **The Main Window** - By default this region is used for the graphic display of the model. It can also be used to display other information, when desired.

- **The Project/Case Manager tree control** - The upper right hand window always contains the Project/Case Manager tree control.
- **The Corner Window** (technically the lower right corner window, however it is referred to elsewhere in boaGui as simply the Corner Window) - By default this region displays the Job Monitoring graph. It can be used to display other information when desired.


Title bar
Menu bar
Project/Case Manager
BoaGui - [Jlab Klystron Gun Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Main Window (defaults to graphic display of model)
x
z
y
Corner Window (defaults to Job Monitoring graph)
Status bar

Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic Beam
Date Modified: Nov 04 2009, 11:51 AM
Comments: User Manual Example 1A - Electrostatic analy

Case: Example 1A
Iteration: 9
Current: 1.46826

FIGURE 3. Main BoaGui Screen

Prior to opening a Case (described in the next section), only the Project/Case Manager section will contain information. It will display a tree control that shows the name of the open Project along with all of the Cases contained within it. Right-clicking on the Project name or on one of the Case names will bring up menus that allow you to manage your Cases and Project (e.g., open a Case, add a new Case to a Project, etc.) Once you open a Case with a defined geometry model, the model will appear in the Main Window. When you execute a run, the Corner Window will display a graph that monitors parameters for the job. These job parameters are specified via the *Execute->Job Parameters* menu.

The pulldown menus at the top of the screen control most of the functions in BoaGui. The remainder of this chapter describes the basic function of each menu item as well as the menu items associated with the Project/Case Manager.

The subsequent chapter contains several example tutorials. The first time user may find it easiest to get familiar with the software by working through the examples first. (See Section 4.0 “Examples” on page 100.)

3.3 Project/Case Manager

The Project/Case Manager appears in the upper right hand window of the screen whenever a Project is open. It consists of a tree control containing all cases in the Project. You can use the tree control to manage the Project and Cases.

3.3.1 Project Menu

Right-clicking on the Project name bring ups the menu shown in Figure 4. It contains the following menu items:

Add New Case - Creates a new Case within the current Project. A dialog box will prompt you to specify the *Name* of the Case, the *Analysis Type*, and any desired *Comments*. You will also have the option to open the case immediately (make it the current case). Once the new Case is created, it will automatically appear beneath the Project in the Project/Case Manager tree control. Behind the scenes, a new Case folder will be placed in the current Project folder. The case folder will include all files specific to the case.

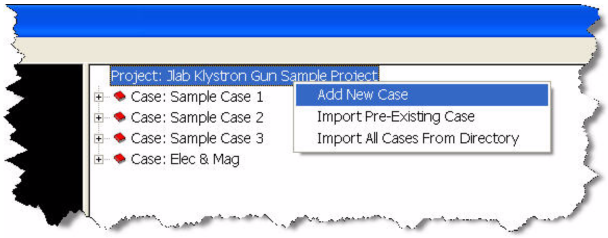


FIGURE 4. Project/Case Manager shown with Project Menu opened

Import Pre-Existing Case - Allows you to import an existing Case into the current Project. You will be prompted to browse the directories on your computer to locate the desired Case folder:

- If you specify a ‘removed’ Case folder in the current Project, the Case will be re-imported into the Project. (A removed Case is one that exists in the Project folder but that has been ‘removed’ from appearing in the Project/Case Manager via the *Remove Case* menu item.)
- If you specify a Case folder in another Project, the Case folder will be copied to the current Project folder. (The case is not moved, it is simply copied. The original case remains located in the original Project folder.)

The imported Case automatically appears beneath the Project in the Project/Case Manager tree control.

Import All Cases From Directory - Allows you to import all Cases from a Project into the current Project. You will be prompted to browse the directories on your computer to locate the desired Project folder:

- If you specify the current Project folder, all ‘removed’ cases will be re-imported into the Project.
- If you specify another Project folder, all Case folders in the Project (including ‘removed’ Cases) will be copied to the current Project. (Cases are copied, not moved. The original cases remain located in the original Project folder.)

All imported Cases automatically appear beneath the Project in the Project/Case Manager tree control.

3.3.2 Case Menu

Right-clicking on one of the Cases brings up the menu shown in Figure 5. It contains the following menu items:

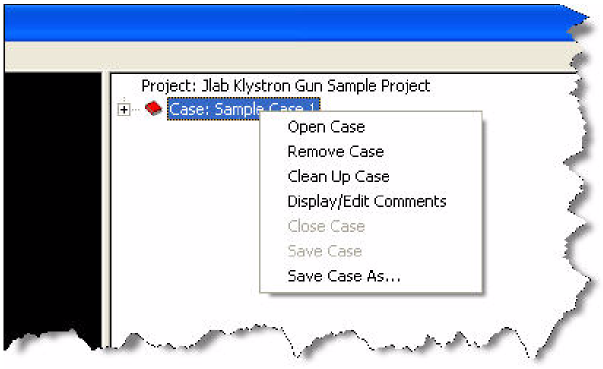

Project: Jlab Klystron Gun Sample Project
Case: Sample Case 1
Open Case
Remove Case
Clean Up Case
Display/Edit Comments
Close Case
Save Case
Save Case As...

FIGURE 5. Project/Case Manager shown with Case Menu opened

Open Case - Opens the selected Case.

Remove Case - Removes the selected Case from the current Project. You will be given two options:

- *Remove the Case from Project and Delete the Case Directory and all of its Contents* - This option deletes the Case directory completely.
- *Remove the Case from Project* - The case will remain in the Project directory but it will not be accessible via the Project/Case Manager. With this option the Case can still be re-imported, but its name cannot be reused for a new case.

Clean Up Case - Deletes files created by a previous BOA run of the case. Deleted files include:

- *.ssv
- *.sss
- *.pws
- *.ebm
- *.out
- *.cde
- *.tdb
- *.fef (Note: The clean up feature removes all *.fef files generated by the case, but does not remove those imported into the case, such as those generated by another case)
- *.jc
- *.trj
- *.msd
- *.mss
- *.spd
- *.res
- *.sle

(See Section 6.3 “Appendix C - Internal Files” on page 183 for a description of all files created by Boa.)

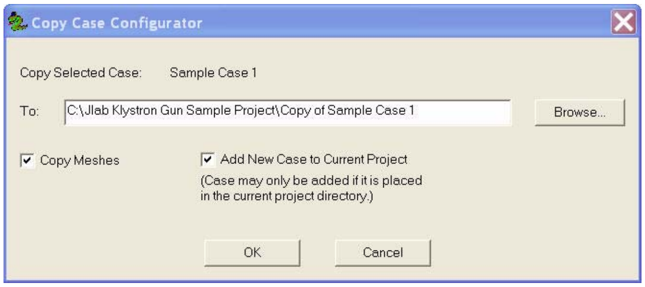
{TBD: check appendix. i think some files were added so we need to mention here}

Display/Edit Comments - Allows you to view and edit the comments for the selected case.

Close Case - Closes the current Case. If you have made any changes to the Case you will be given the option to save the Case or close without saving.

Save Case - Saves the current Case.

Save Case As... - Allows you to copy the selected case to either the current Project directory (with a different Case name) or another Project directory (with the same or different Case name). This can be useful for backing up cases and can also be useful for sending a case to CCR for debugging during the beta test period. Figure 6 shows the Copy dialog.



Copy Case Configurator

Copy Selected Case: Sample Case 1

To: C:\Jlab Klystron Gun Sample Project\Copy of Sample Case 1

☑ Copy Meshes

☑ Add New Case to Current Project

(Case may only be added if it is placed in the current project directory.)

FIGURE 6. Copy Case dialog

- If you copy the case to the current Project directory, the *Add New Case to Current Project* checkbox gives you the option to ‘add’ the case to the Project. (Adding the case means that in addition to it being copied into the directory it will also appear in the Project/Case Manager.) If you choose not to add the case when you copy, you can always add it later using the *Import Pre-Existing Case* feature.
- If you copy the case to another Project directory, the case will be copied only -- not added. You will need to use the *Import PreExisting Case* feature (when that Project is open) in order for it to appear in the Project Case Manager.

The *Copy Meshes* checkbox gives you the option to include or exclude the meshes when the copy is performed.

3.4 File Menu

The **File** pulldown menu is shown in Figure 7. It includes the following:

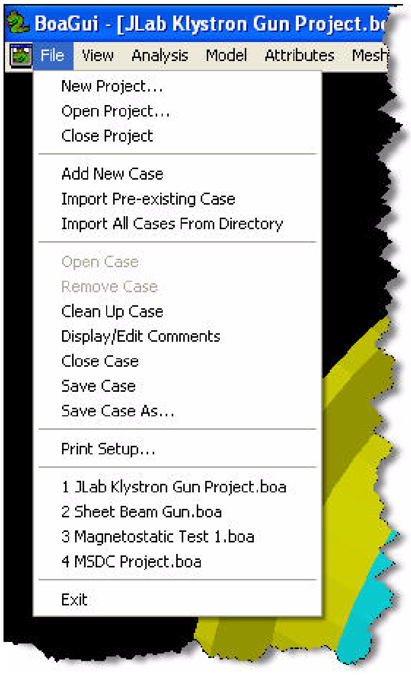

BoaGui - [JLab Klystron Gun Project.bo
File
View
Analysis
Model
Attributes
Mesh
New Project...
Open Project...
Close Project
Add New Case
Import Pre-existing Case
Import All Cases From Directory
Open Case
Remove Case
Clean Up Case
Display/Edit Comments
Close Case
Save Case
Save Case As...
Print Setup...
1 JLab Klystron Gun Project.boa
2 Sheet Beam Gun.boa
3 Magnetostatic Test 1.boa
4 MSDC Project.boa
Exit

FIGURE 7. File Pulldown Menu

New Project - Opens a new Project. BoaGui will prompt you to specify the name of the Project and the directory in which you would like the new Project folder to reside. The Project folder will contain the Project file (*.boa) and all associated Case folders.

Open Project - Opens an existing Project. You will be prompted to browse the directories on your computer to locate the desired Project file (*.boa).

Close Project - Closes the current Project and Case. If you have made any changes to the Case you will be given the option to save the Case (and Project) or close without saving.

Add New Case - Identical to menu item with the same name on the Project 'right-click' menu. See previous section for details.

Import Pre-Existing Case - Identical to menu item with the same name on the Project 'right-click' menu. See previous section for details.

Import All Cases From Directory - Identical to menu item with the same name on the Project 'right-click' menu. See previous section for details.

Open Case - Identical to menu item with the same name on the Case 'right-click' menu. See previous section for details. Use the tree control in the Project/Case Manager to highlight the desired Case.

Remove Case - Identical to menu item with the same name on the Case 'right-click' menu. See previous section for details. If a case is not open, use the tree control in the Project/Case Manager to highlight the desired Case.

Clean Up Case - Identical to menu item with the same name on the Case 'right-click' menu. See previous section for details. If a case is not open, use the tree control in the Project/Case Manager to highlight the desired Case.

Display/Edit Comments - Identical to menu item with the same name on the Case 'right-click' menu. See previous section for details. If a case is not open, use the tree control in the Project/Case Manager to highlight the desired Case.

Close Case - Identical to menu item with the same name on the Case 'right-click' menu. See previous section for details.

Save Case - Identical to menu item with the same name on the Case ‘right-click’ menu. See previous section for details.

Save Case As - Identical to menu item with the same name on the Case ‘right-click’ menu. See previous section for details. If a case is not open, use the tree control in the Project/Case Manager to highlight the desired Case.

Print Setup... - Opens the *Print Setup* dialog box where you can select the printer and various print options.

File History - Displays a history of the past four Project files opened. This allows easy selection of previously loaded files.

Exit - Closes BoaGui. If the currently loaded Case has not been saved, you will be given the option to save the Case or exit without saving.

Upon initiating BoaGui you must create or open a Project and then create or open a Case. Most pulldown menus will be disabled until Project and Case are specified.

3.5 View Menu

The **View** pulldown menu, shown in Figure 8, contains options that affect the display in the graphics region. The menu items include:

Orientation - Provides predefined orientations for the graphics display, including: *zx (-y)*, *zx (+y)*, yz *(-x)*, *yz (+x)*, *xy (-z)*, *xy (+z)*, and *Isometric*. (The *zx (-y)* orientation represents the zx plane with the -y axis projecting out from the screen, and so on.)

Surface Representation - Provides two types of visualization, *Wireframe* and *Surface*. Examples of both types are shown in Figure 9. *Surface* is the default view. You can also toggle between the two modes by typing the *W* key or the *S* key on the keyboard.

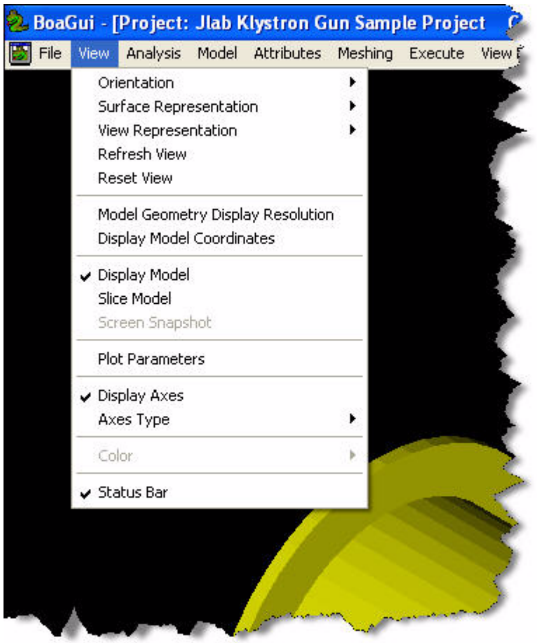

BoaGui - [Project: Jlab Klystron Gun Sample Project
File
View
Analysis
Model
Attributes
Meshing
Execute
View
Orientation
Surface Representation
View Representation
Refresh View
Reset View
Model Geometry Display Resolution
Display Model Coordinates
✓ Display Model
Slice Model
Screen Snapshot
Plot Parameters
✓ Display Axes
Axes Type
Color
✓ Status Bar

FIGURE 8. View pulldown menu

This setting applies to all items shown in the graphics display region. This can include the geometric model, the mesh (initial and adapted), and the resulting electrostatic and magnetic fields.

View Representation - Provides two options for view representation, *Orthogonal* and *Perspective*. Examples of each type are shown in Figure 10. *Orthogonal* is the default representation. It displays the ‘true’ view (view rays are parallel and objects are rendered without distance effect). It provides accurate distance measurements when using the *Measure Distance* feature under the **View Results** menu.

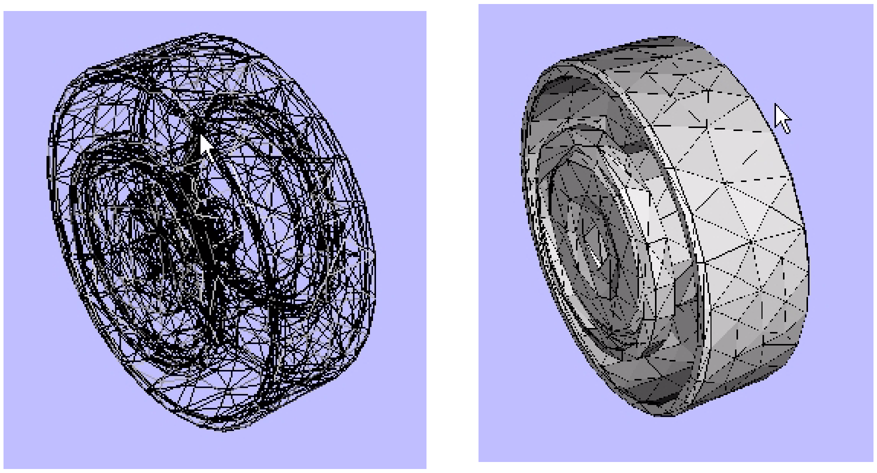


FIGURE 9. Examples of surface representations. A *Wireframe* representation is shown on the left and a *Surface* representation is shown on the right.

The *Perspective* representation provides a more pleasing view in three dimensions, however it distorts the position of objects relative to each other. (View rays are NOT parallel and objects are rendered more natural looking but distortion is introduced.) WARNING: The *Perspective* representation should never be used with the *Measure Distance* feature. It will result in incorrect distance measurements. {*TBD: Check if this is still true. this is a very early constraint Bill Vogler gave me; if it is true, can we prohibit this in the GUI? if not, then remove this comment.*}

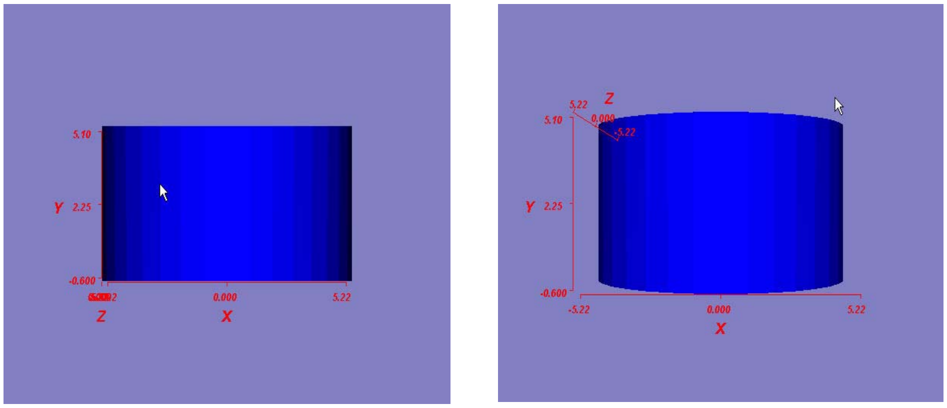


FIGURE 10. Examples of a view representations. The left image shows an *Orthogonal* representation of a cylinder. The right image shows a *Perspective* representation of the same cylinder.

Experiment with both representations to determine which option you prefer for general viewing and printing. However, always use the *Orthogonal* representation when using the *Measure Distance* feature.

Refresh View - Refreshes the display such that the current orientation remains fixed but the entire model is visible and centered within the graphics display window. (This can be used to quickly view the whole model after zooming in on a specific region.) Typing the *R* key will also refresh the display.

Reset View - Resets the display to the isometric view. (This command is identical to the *Isometric* item on the *File-> Orientation* menu.)

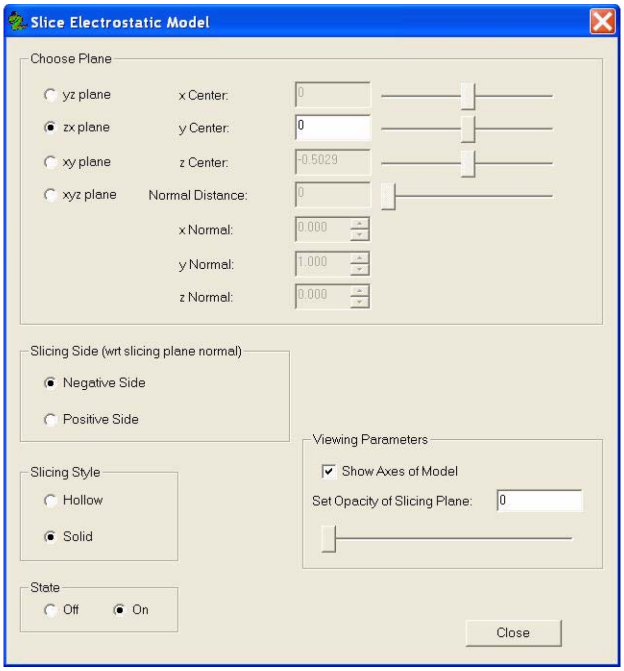
Model Geometry Display Resolution - Controls the *Relative Resolution* display of curved surfaces. Curved surfaces are actually composed of sections of flat surfaces. With a higher resolution, more flat surfaces are used to render the curved surface and the surface appears smoother. The *Relative Resolution* can be set between zero and one, with one being the highest resolution and zero being the lowest. The higher the resolution, the longer it takes for the computer to generate the initial display, but manipulation of the display thereafter (rotation, zoom, pan) is not significantly affected. The default resolution is set to 0.5.

NOTE: For this parameter to take effect, you must save and close the case and then reopen it.

Display Model Coordinates - Dynamically displays the X, Y, Z coordinates of the mouse cursor as it moves through the graphics display region. When this feature is on, a check mark appears in front of the menu item and the three coordinates appear in the status bar.

Display Model - Shows and hides the geometric model. (Hiding the model can sometimes be useful when viewing results.) When the display is on, a check mark appears in front of the menu item.

Slice Model - This menu item allows you to create a slicing plane and remove the model on either side of the plane, providing a view of the interior of the model. This can be useful when assigning attributes or when viewing results (for example, when verifying that a beam is centered in a drift tunnel or to measure beam minimum position and diameter inside a part).


Slice Electrostatic Model

Choose Plane
○ yz plane x Center: 0
● zx plane y Center: 0
○ xy plane z Center: -0.5029
○ xyz plane Normal Distance: 0
x Normal: 0.000
y Normal: 1.000
z Normal: 0.000

Slicing Side (wrt slicing plane normal)
● Negative Side
○ Positive Side

Slicing Style
○ Hollow
● Solid

State
○ Off ● On

Viewing Parameters
☑ Show Axes of Model
Set Opacity of Slicing Plane: 0

FIGURE 11. Slicing Input Parameters dialog box

The menu item brings up the dialog box shown in Figure 11. The *Choose Plane* radio buttons define the plane used to slice the model. Choose one of the three standard orthogonal cut planes (*xy*, *xz* or *yz plane*) or an arbitrary cut plane (*xyz plane*). For the arbitrary cut plane, specify the three components of the vector normal to the plane: *x Normal*, *y Normal* and *z Normal*. (For example, the positive normal vector to the xy plane is <0, 0, 1>, and the negative normal vector is <0, 0, -1>.) Use the slider bars to define the distance of the plane along the normal vector.

The *Slicing Side* radio buttons control which side of the plane is removed. The *Positive Side* refers to the side normal to the plane and the *Negative Side* refers to the opposite side. The *Slicing Style* radio buttons provide options for the appearance of the sliced edge of the model (*Hollow* or *Solid*). The *Viewing Parameters* section contains a checkbox that shows/hides the axes and a slider bar that controls the opacity of the slicing plane. The *Slicing State* radio button lets you turn the feature on and off (i.e., show or hide the sliced portion of the model). When the radio button is set to *On*, a check mark appears in front of the *Slice Model* menu item on the *File* menu.

An example of a sliced model is shown in Figure 12.

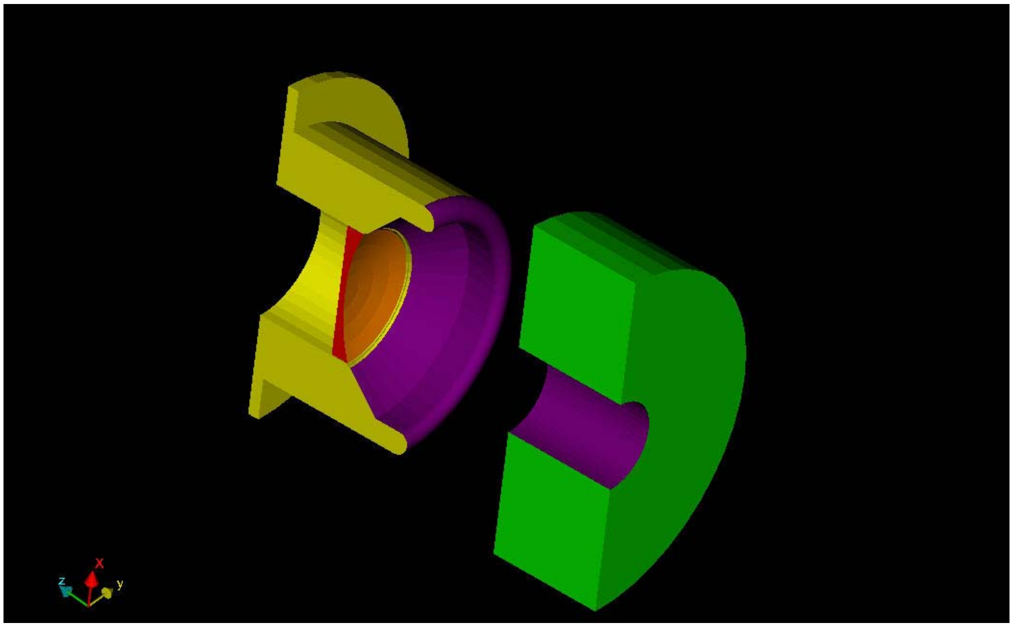


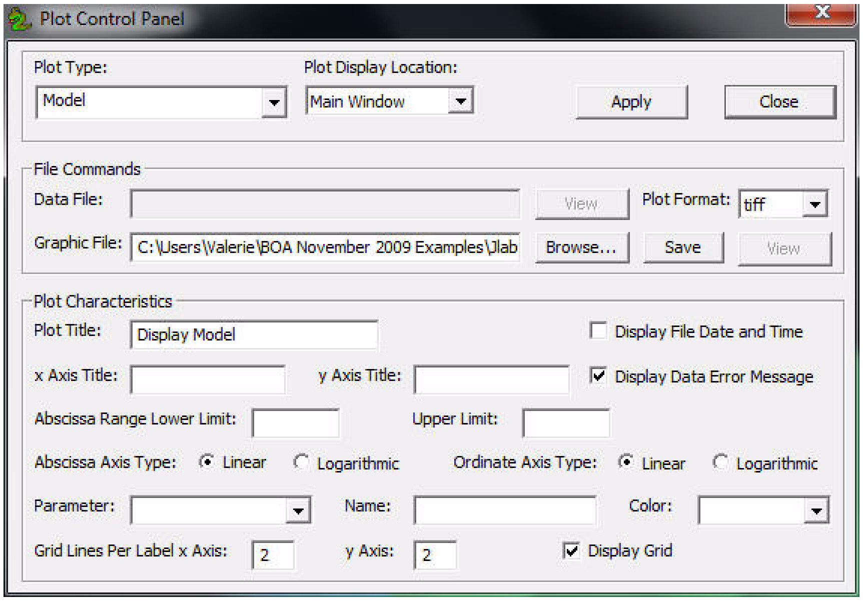
FIGURE 12. Example of Slicing the Model

Screen Snapshot - This feature saves a snapshot of the Main Window (which typically displays the geometry model) to a file named ‘MainScreen.jpg’ and puts it in your case directory.

Plot Parameters - This menu item brings up the *Plot Control Panel* dialog shown in Figure 13. Use this dialog to view various plots and specify where they appear.

The *Plot Type* drop down box contains the following options:

- Convergence *(a line plot)*
- Permeability *(a line plot)*
- Particle *(a scatter plot)*
- Electrostatic Line Plot *(a line plot)*



Plot Control Panel

Plot Type: Model

Plot Display Location: Main Window

Data File:

Plot Format: tiff

Graphic File: C:\Users\Valerie\BOA November 2009 Examples\Jlab

Plot Title: Display Model

☐ Display File Date and Time

x Axis Title:

y Axis Title:

☑ Display Data Error Message

Abscissa Range Lower Limit:

Upper Limit:

Abscissa Axis Type: ● Linear ○ Logarithmic

Ordinate Axis Type: ● Linear ○ Logarithmic

Parameter:

Name:

Color:

Grid Lines Per Label x Axis: 2

y Axis: 2

☑ Display Grid

FIGURE 13. Plot Control Panel dialog

- Magnetostatic Line plot *(a line plot)*
- External Magnetic Field *(a line plot)*
- Model *(the geometric model)*
- Optimizer Convergence *{get more info}*
- Error Estimates *{get more info}*
- Newton Residual *{get more info}*

{This section currently under revision;

- *More details to be added. integrate (or show) all info from my Excel table summary here*
- *confirm bugs i reported were fixed}*

Display Axes - Shows and hides the XYZ axes along with coordinates enclosing the displayed electrostatic part(s). When the display is On, a check mark appears in front of the menu item. Figure 14 shows an example of the axes displayed with Axes Type = 'Box Style' (as explained below). Figure 12 shows an example of the axes displayed with Axes Type = 'CAD Style'.

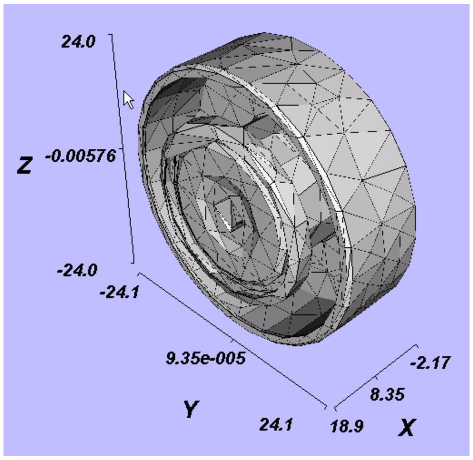


FIGURE 14. Example of Axes Displayed (with Axis Type = Box Style)

Axes Type - BoaGui provides two options for the appearance of the axes: 'CAD Style' and 'Box Style'. CAD Style is the default. Box Style shows a box enclosing the model and displays the axis coordinates. The default Axes Type is CAD Style. Figure 14 shows an example of the Axes Type set to Box Style.

Color - Opens the submenu shown in Figure 15. The last item, *Trajectories*, will be active only if beam analysis results have been generated. Selecting any of the first three items (*Background*, *Axes*, and *Mesh*) results in the *Color* palette dialog shown in Figure 16. Use this dialog to set the desired color for the *Background*, the *Axes*, or the *Mesh*.

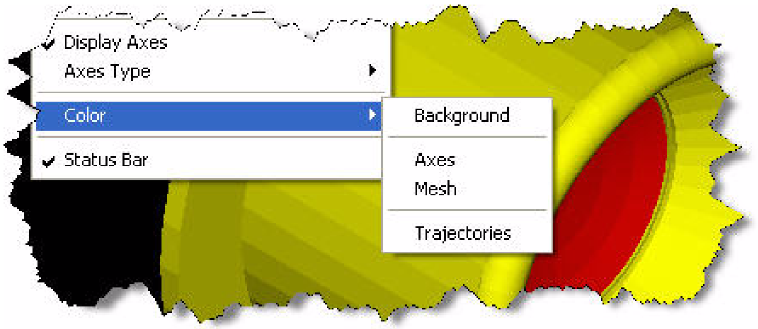

✓ Display Axes
Axes Type
Color
✓ Status Bar
Background
Axes
Mesh
Trajectories

FIGURE 15. Color Submenu

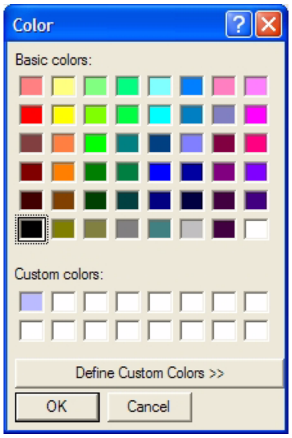
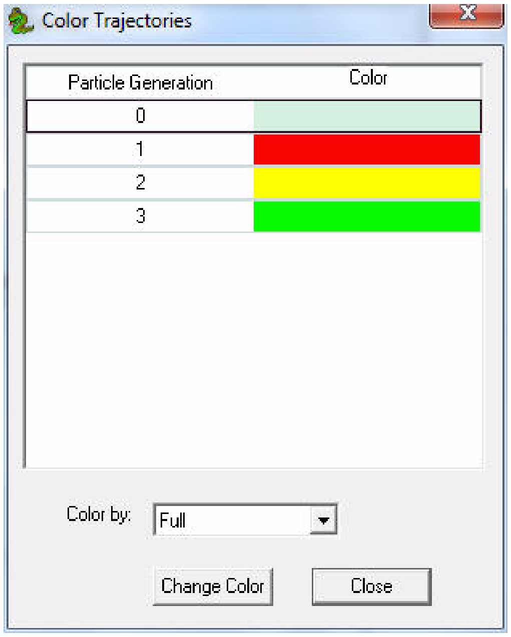

Color
Basic colors:
Custom colors:
Define Custom Colors >>
OK
Cancel

FIGURE 16. Color Palate dialog


Color Trajectories
Change Color
Close

Particle Generation	Color
0	
1	🟥
2	🟨
3	🟩

Color by: Full

FIGURE 16. Color Trajectories dialog

The *Trajectories* item opens the dialog shown in Figure 16. To change the color of the trajectories for a given Particle Generation, select the Particle Generation and click the *Change Color* button. The Color Palette dialog shown in Figure 16 will appear. The *Color by* dropdown box allows you to further control the color of the trajectories as follows:

‘Full’ - shows the selected single color for all trajectories.

‘Particle ID’ - varies the color of each trajectory based on its Particle ID. The Particle ID legend pops up at the bottom of the Main Window when this option is selected.

‘Particle Current’ - varies the color of each trajectory based on its current. The Particle Current legend pops up at the bottom of the Main Window when this option is selected.

Status Bar - Shows and Hides the status bar at the bottom of the boaGui screen. When the status bar is shown, a check mark appears in front of the Status Bar menu item.

Zoom Feature - BoaGui also includes a *Zoom* feature although it is not menu driven. The right mouse button controls the *Zoom* feature. To zoom in, hold down the right mouse button and move the mouse forward (or ‘up’). To zoom out, hold down the right mouse button and move the mouse backward (or ‘down’).

3.6 Analysis Menu

The **Analysis** pulldown menu, shown in Figure 17, sets the analysis type for the current case. The selected analysis type appears preceded by a check mark. Each analysis type requires different input parameters. BoaGui enables and disables the appropriate menus and execution parameters based on this selection. Options include:

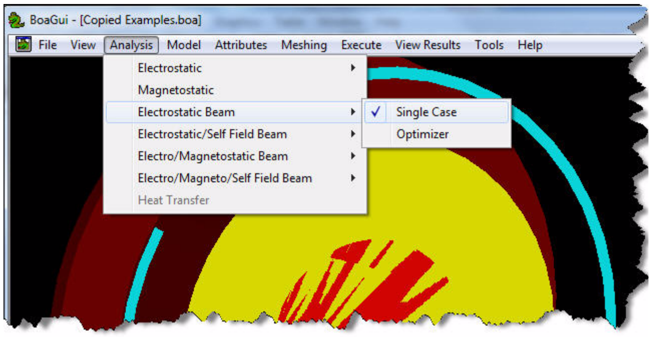

BoaGui - [Copied Examples.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Electrostatic
Magnetostatic
Electrostatic Beam
Electrostatic/Self Field Beam
Electro/Magnetostatic Beam
Electro/Magneto/Self Field Beam
Heat Transfer
✓ Single Case
Optimizer

FIGURE 17. Analysis pulldown menu

- **Electrostatic** - Performs an electrostatic analysis only.
- **Magnetostatic** - Performs a magnetostatic analysis only.
- **Electrostatic Beam** - Performs an electrostatic analysis, followed by a beam simulation, updating and refining the electric fields based on space charge effects. Self magnetic field from the electron beam is not included in the simulation.
- **Electrostatic/Self Field Beam** - Performs an electrostatic analysis, followed by a beam simulation, updating and refining the electric fields based on space charge effects. Self magnetic field from the electron beam is included in the simulation.
- **Electro/Magnetostatic Beam** - Performs an electrostatic simulation, followed by a magnetostatic simulation. The magnetostatic fields are interpolated to the electrostatic mesh during subsequent iterations. The electron beam is simulated and the electric fields are updated based on space charge effects. Self magnetic field from the beam electrons is not included.
- **Electro/Magneto/Self Field Beam** - Performs an electrostatic simulation, followed by a magnetostatic simulation. The magnetostatic fields are interpolated to the electrostatic mesh during subsequent iterations. The electron beam is simulated and the electric fields are updated based on space charge effects. Self magnetic field from the beam electrons is included.
- **Heat Transfer** - A thermomechanical modeling module is planned for future versions. This feature is not available in the initial beta release.

All Electrostatic analysis types include the option for a **Single Case** analysis or an **Optimizer Analysis**:

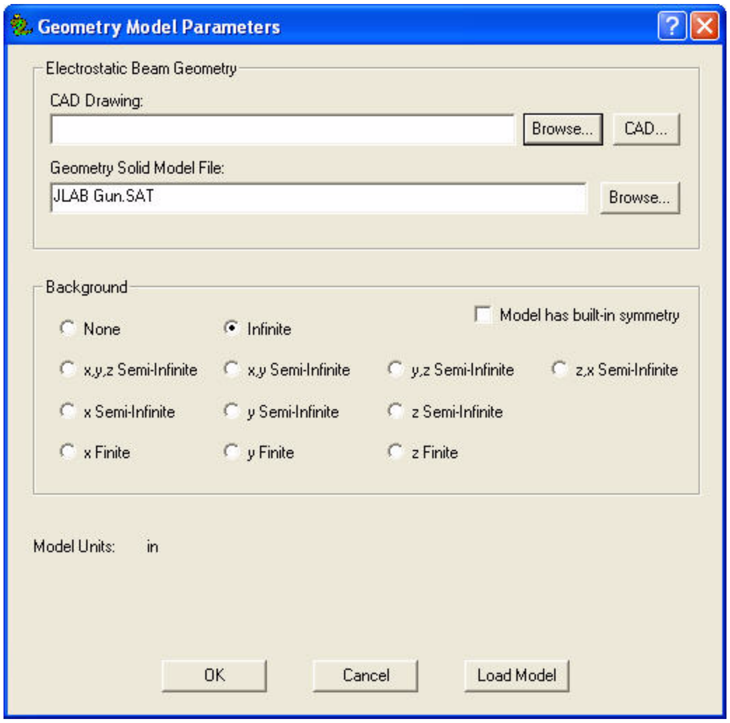
- Executing a Single Case means the user imports a single geometry, specifies a single set of problem attributes, meshes and then executes for that geometry and attribute set.
- In contrast, an Optimizer Analysis allows the user define goals for the optimal design, then based on these goals, Boa runs, updates the geometry and other problem attributes, re-meshes and reruns until the goals are achieved.

Note: Running the Optimizer requires both Excel and SolidWorks. BOA will issue an error if the system does not meet these requirements.

During an Optimization run, the lower Corner Window will display the progress of the optimization.

3.7 Model Menu

{TBD: This section needs some revisions: add discussion of the multiple information messages that appear and explain initial vs. final model. (I already added this to the examples but did not yet add here.) Also, update section referenced below.}



Geometry Model Parameters
Electrostatic Beam Geometry
CAD Drawing:
Browse...
CAD...
Geometry Solid Model File:
Browse...
Background
○ None
● Infinite
☐ Model has built-in symmetry
○ x,y,z Semi-Infinite
○ x,y Semi-Infinite
○ y,z Semi-Infinite
○ z,x Semi-Infinite
○ x Semi-Infinite
○ y Semi-Infinite
○ z Semi-Infinite
○ x Finite
○ y Finite
○ z Finite
Model Units:
OK
Cancel
Load Model

Geometry Solid Model File: JLAB Gun.SAT

Background: ● Infinite

Model Units: in

FIGURE 18. Geometry Model Parameters dialog

The **Model** menu item opens the *Geometry Model Parameter*s dialog box, shown in Figure 18. It contains two main sections, one for specifying *Geometry* and one for specifying *Background*. The *Model Units* are shown at the bottom of the dialog.

The exact title of the *Geometry* section varies with the selected *Analysis Type*. (For example, in Figure 18 it is titled *Electrostatic Beam Geometry* because the *Analysis Type* for this particular analysis is set to ‘Electrostatic Beam’.) The *Geometry* section contains the following:

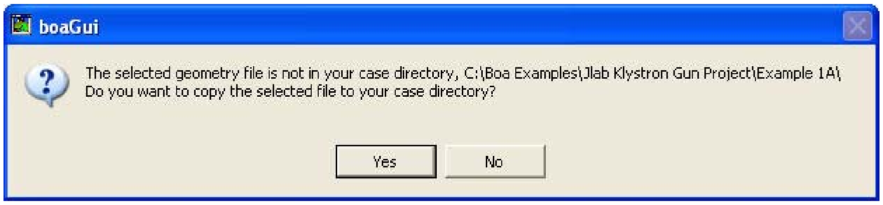
CAD Drawing - This feature currently works for CAD drawings created in SolidWorks only:

- Optimization run: The *CAD Drawing* is required for an Optimization run. The drawing is updated with each consecutive optimization run and a new geometry file is imported into the case to meet the specified optimization goals. This CAD drawing must be created in SolidWorks and SolidWorks is a system requirement for Optimization runs.
- Single case - For a single case run, this feature is provided for convenience only. It allows you to access the SolidWorks CAD drawing used to create the geometry file without leaving BoaGui. *WARNING:* If you make a change to the CAD drawing in SolidWorks, BOA will not automatically copy the modified geometry file into the Case directory. Make sure to copy the modified file to the Case directory by re-specifying it as described in the section below.

Use the *Browse* button to locate (specify path for) the desired CAD file. Use the *CAD* button to access the specified CAD drawing in SolidWorks.

Geometry Solid Model File - Used to specify the geometry file (either an ACIS *.sat file or a Para-Solid *.x_t file) on which the analysis is based. This file is always required.

*Note: for the current beta release the ParaSolid *.x_t file format has not been fully tested due to the issue with model units always in meters.*



The selected geometry file is not in your case directory, C:\Boa Examples\Jlab Klystron Gun Project\Example 1A\
Do you want to copy the selected file to your case directory?

FIGURE 19. Geometry File Copy Message

Use the *Browse* button to locate the desired file. When you select the file, BOA will display the message shown in Figure 19 informing you that the specified file will be copied to the Case directory. Press *Yes* to proceed. (BoaGui requires that a copy of the geometry file reside in the BOA case directory. BoaGui does not use the original version of geometry file.)

Note that subsequent steps in a BOA analysis depend on the geometry model file, including attribute assignments and mesh generation. If you specify a new geometry file, BOA will issue a Warning message with three options:

- *Update* - updates boundary and source assignments. To successfully update, the new geometry model must have the same number of parts and the same part names as the old model.
- *New* - automatically clears attribute assignments and deletes the existing mesh file so that you can start new assignments
- *Cancel*

See the Appendix for details on how parts and surfaces should be named within the geometry file.

The *Background* section *{TBD: provide explanation; waiting for implementation of redesign...}*

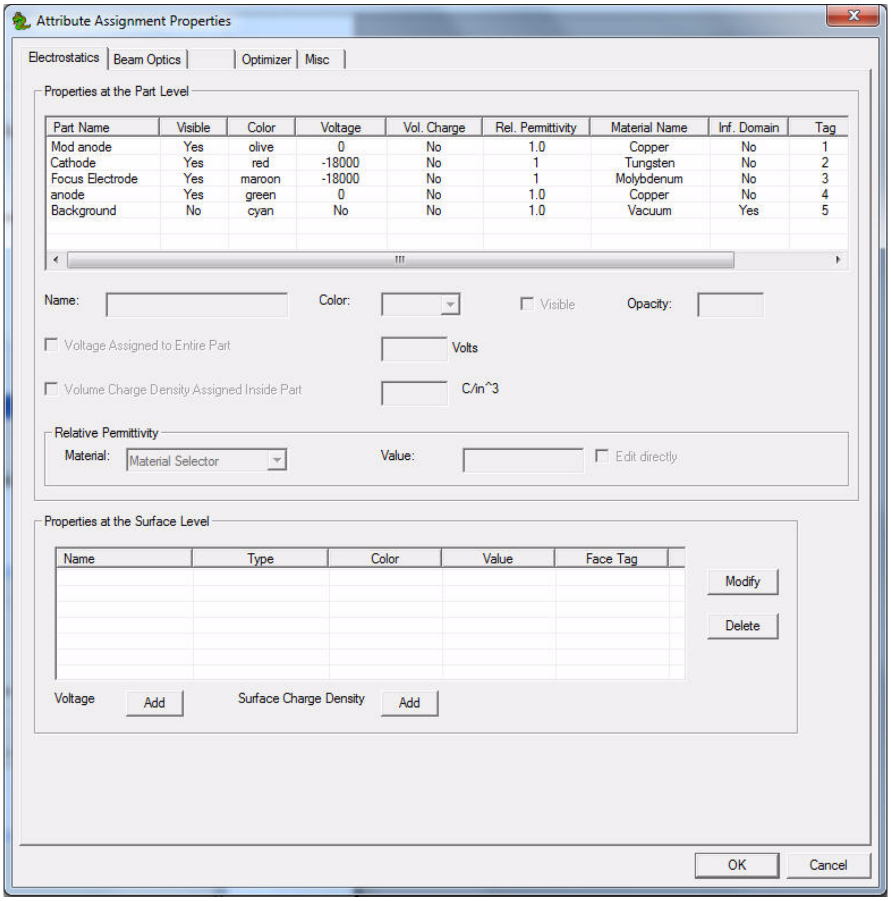
Model Units - These units are taken directly from the geometry model file.

3.8 Attributes Menu

The **Attributes** menu item brings up the tabbed *Attribute Assignment Properties* dialog shown in Figure 20. It contains five possible tabs: *Electrostatics, Beam Optics*, *Magnetostatics*, *Optimizer* and *Miscellaneous*. The tabs vary depending on the selected analysis type. For example, in Figure 20, the *Magnetostatic* tab is absent because the **Analysis** menu is set to ‘Electrostatic Beam’. Note that the *Miscellaneous* tab is for internal CCR use during the Beta testing period only. Users should not use this tab unless troubleshooting with CCR personnel.

3.8.1 Electrostatic Attributes

The *Electrostatics* tab, shown in Figure 20, is used to assign properties required for an electrostatic analysis. It is enabled when the **Analysis** menu is set to one of the Electrostatic analysis types. The tab includes sections for defining properties at both the part level and the surface level.


Attribute Assignment Properties
Electrostatics
Beam Optics
Optimizer
Misc
Properties at the Part Level
Name:
Color:
☐ Visible
Opacity:
☐ Voltage Assigned to Entire Part
Volts
☐ Volume Charge Density Assigned Inside Part
C/in^3
Relative Permittivity
Material:
Material Selector
Value:
☐ Edit directly
Properties at the Surface Level
Name
Type
Color
Value
Face Tag
Modify
Delete
Voltage
Add
Surface Charge Density
Add
OK
Cancel

Part Name	Visible	Color	Voltage	Vol. Charge	Rel. Permittivity	Material Name	Inf. Domain	Tag
Mod anode	Yes	olive	0	No	1.0	Copper	No	1
Cathode	Yes	red	-18000	No	1	Tungsten	No	2
Focus Electrode	Yes	maroon	-18000	No	1	Molybdenum	No	3
anode	Yes	green	0	No	1.0	Copper	No	4
Background	No	cyan	No	No	1.0	Vacuum	Yes	5

FIGURE 20. Attribute Assignment dialog box, Electrostatics tab

3.8.1.1 Properties at the Part Level

The *Properties at the Part Level* section allows you to assign properties to the parts that make up the geometric model. The table at the top of the section contains all of the parts defined in the electrostatic geometry file. It is automatically populated based on the file specified on the *Geometry Model Parameters* dialog. Additional entries cannot be added. To modify a part's properties, first select the part to make it active. The properties beneath the table will then become available. Properties include:

Name - The part name is identical to the name provided in the geometry file. This property is not editable.

Color - Specify a color for the part from the *Color* dropdown box.

Visible checkbox - Check/uncheck the box to show/hide the part in the graphics display of the geometry model (shown in the Main Window). BoaGui will initially show all parts -- except for the background part -- as visible.

Opacity - *Opacity* is the translucency of a part for the purposes of graphical display. It ranges from 1 to 0, going from no transmission of light through the part (opaque) to full light transmission (transparent). A part's *Opacity* is editable only if it is set to *Visible*. (Notice that although by default the background part is set to *invisible*, if you change it to *visible* then its *Opacity* defaults to 0.8. This default value allows you to see through it to the internal parts.)

Voltage Assigned to the Entire Part checkbox - Check the box for any part that has potential and enter the voltage in Volts. If the box is left unchecked, the part is assumed to have no potential. (An unchecked box is not the same as a checked box with a voltage of zero.)

Volume Charge Density Assigned Inside Part - To assign a volume charge density inside of the part, check the box and enter the volume charge density in Coloumbs per cubic ‘length unit’ (the length unit (e.g., inch) is defined in the geometry file and shown on the *Geometry Model Parameters* dialog). If the box is left unchecked, the part is assumed to have no volume charge density.

Relative Permittivity section - The *Relative Permittivity* for a part can be entered directly or looked up in the *Materials Properties* database. (See Section 3.12.3 for a description of the *Materials Properties* database.) Typically it is looked up, so by default the *Value* field is non-editable (i.e., the *Edit directly* checkbox next to it is unchecked) and the *Material* dropdown box is used to find the *Relative Permittivity.*

The *Edit Directly* checkbox allows you to enter a *Relative Permittivity Value* directly. Use this feature if you want to either:

- enter a value independent of the *Material Properties* database (the material name will default to ‘Noname’ when it appears in the Parts table), or
- edit the *Relative Permittivity Value* of a user-provided material in the *Material Properties* database.

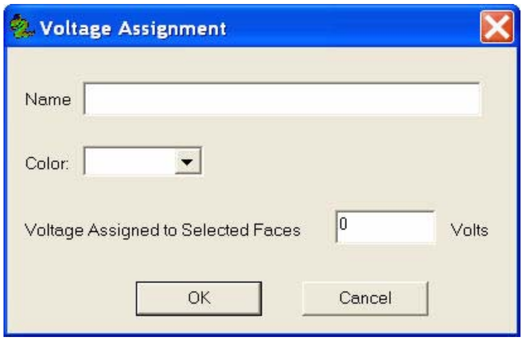
You can also add materials to the *Material Properties* database by selecting ‘Add Your Own Material’ in the *Material* drop-down box. This will bring up the *Material Properties* dialog discussed in Section 3.12.3. (It can also be accessed via the *Tools->Material Properties* menu.)

3.8.1.2 Properties at the Surface Level

The *Properties at the Surface Level* section (shown in Figure 20) is used to assign voltages and surface densities to specific surfaces in the model. Unlike *Properties at the Part Level,* the Surface Property table is empty until you explicitly define properties to specific surfaces.

> *{TBD: Decide if this belongs here or in an earlier, more general discussion about surfaces throughout the BOA GUI: Each part in the geometry model consists of multiple surfaces. These surfaces are defined when the CAD model is created. Surfaces in the geometry model can be named in the CAD program that generates the geometry model. If the geometry model contains names for the defined surfaces, BoaGui will retain these names and will not allow you to edit the names. If the geometry model contains surfaces that have not been named, BoaGui will prompt you to name any surface for which you define properties. These names will be editable.}*

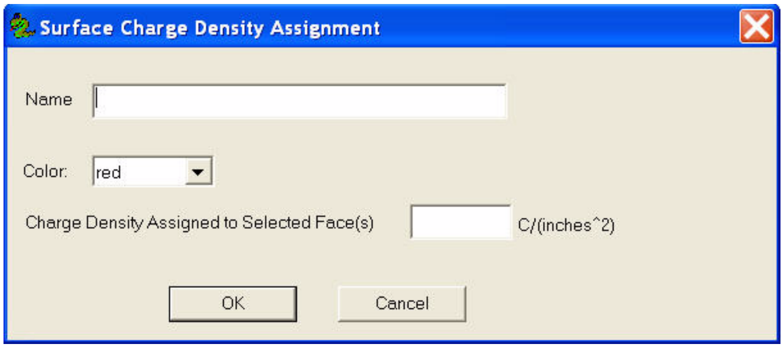
To assign a voltage to a surface, click the *Voltage Add* button beneath the table. This will bring up the Main Window containing the geometric model. Select the desired surface by pressing Ctrl-Left mouse button and pressing *Enter*. The *Voltage Assignment* dialog box in Figure 21 will appear. Enter the desired *Name* and *Voltage* and specify the *Color*. (If the surface is already named in the geometry model, the *Name* will not be editable.) Click *OK* to complete the assignment and the item will appear in the table.



Voltage Assigned to Selected Faces 0 Volts

FIGURE 21. Surface Voltage Assignment dialog

To assign a surface charge density to a surface, click the *Surface Charge Density Add* button beneath the table. This will bring up the Main Window containing the geometric model. Select the desired surface by pressing Ctrl-Left mouse button and pressing Enter. The *Surface Charge Density Assignment* dialog in Figure 22 will appear. Enter the desired *Name* and *Charge Density* and specify the *Color*. (If the surface is already named in the geometry model, the *Name* will not be editable.)Click *OK* to complete the assignment and the item will appear in the table.


Surface Charge Density Assignment
Name
Charge Density Assigned to Selected Face(s)
C/(inches^2)
OK
Cancel

Color: red

FIGURE 22. Face Surface Charge Density Assignment dialog box

To modify a surface property once it is defined, select the item in the table and click the *Modify* button. The appropriate dialog will appear and you can edit the values. To delete a surface property, select the item in the table and click the *Delete* button.

3.8.2 Beam Optics Attributes

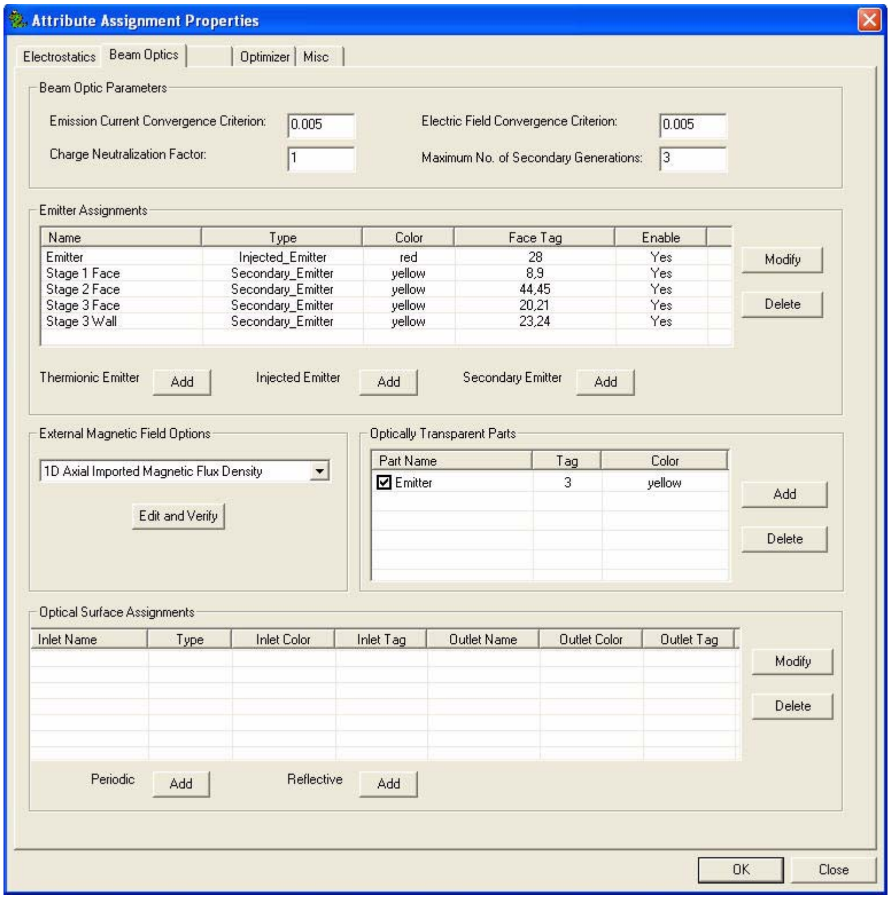
The *Beam Optics* tab, shown in Figure 23, enables you to make the attribute assignments required for beam analyses. The screen includes sections for specifying the following:

- *Beam Optics Parameters*
- Emitter Assignments
- External Magnetic Field
- Optically Transparent Parts
- Optical Surface Assignments (Periodic and Reflective surfaces).

3.8.2.1 Beam Optic Parameters

The *Beam Optic Parameters* section includes the following parameters:

Emission Current Convergence Criteria - This is one of the two convergence criteria for the BOA algorithm. The solution converges if both EQ 1 and EQ 2 are met.


Attribute Assignment Properties
Electrostatics
Beam Optics
Optimizer
Misc
Beam Optic Parameters
Emitter Assignments
Modify
Delete
Add
External Magnetic Field Options
Edit and Verify
Optically Transparent Parts
Optical Surface Assignments
Periodic
Reflective
OK
Close

Emission Current Convergence Criterion: 0.005
Electric Field Convergence Criterion: 0.005
Charge Neutralization Factor: 1
Maximum No. of Secondary Generations: 3

Name	Type	Color	Face Tag	Enable
Emitter	Injected_Emitter	red	28	Yes
Stage 1 Face	Secondary_Emitter	yellow	8,9	Yes
Stage 2 Face	Secondary_Emitter	yellow	44,45	Yes
Stage 3 Face	Secondary_Emitter	yellow	20,21	Yes
Stage 3 Wall	Secondary_Emitter	yellow	23,24	Yes

Thermionic Emitter Add
Injected Emitter Add
Secondary Emitter Add

1D Axial Imported Magnetic Flux Density

Part Name	Tag	Color
☑ Emitter	3	yellow

Inlet Name	Type	Inlet Color	Inlet Tag	Outlet Name	Outlet Color	Outlet Tag

FIGURE 23. Attribute Assignment dialog, Beam Optics tab

$$\left| 1 - \frac{\text{Emission Current}_{\text{previous iteration}}}{\text{Emission Current}_{\text{curre nt iteration}}} \right| < \mathit{Emission\ Current\ Convergence\ Criteria} \qquad \text{(EQ 1)}$$

The default value is 0.005.

Potential Convergence Criteria - This is the second convergence criterion for the BOA algorithm. The solution converges if both EQ 1 and EQ 2 are met.

$$\left| 1 - \frac{\text{Maximum Potential}_{\text{previous iteration}}}{\text{Maximum Potential}_{\text{curre nt iteration}}} \right| < \mathit{Potential\ Convergence\ Criteria} \qquad \text{(EQ 2)}$$

The default value is 0.005.

Charge Neutralization Factor - This parameter accounts for charge neutralization (ionic charges neutralizing electronic charges). Each particle charge is reduced by this factor. This value must be less than or equal to 1. The default value is 1 (i.e., no charge neutralization).

Maximum Number of Secondary Generations - This parameter is enabled only when a secondary emitter is defined in the *Emitter Assignments* section. It defines the maximum number of possible secondary generations per one primary electron.

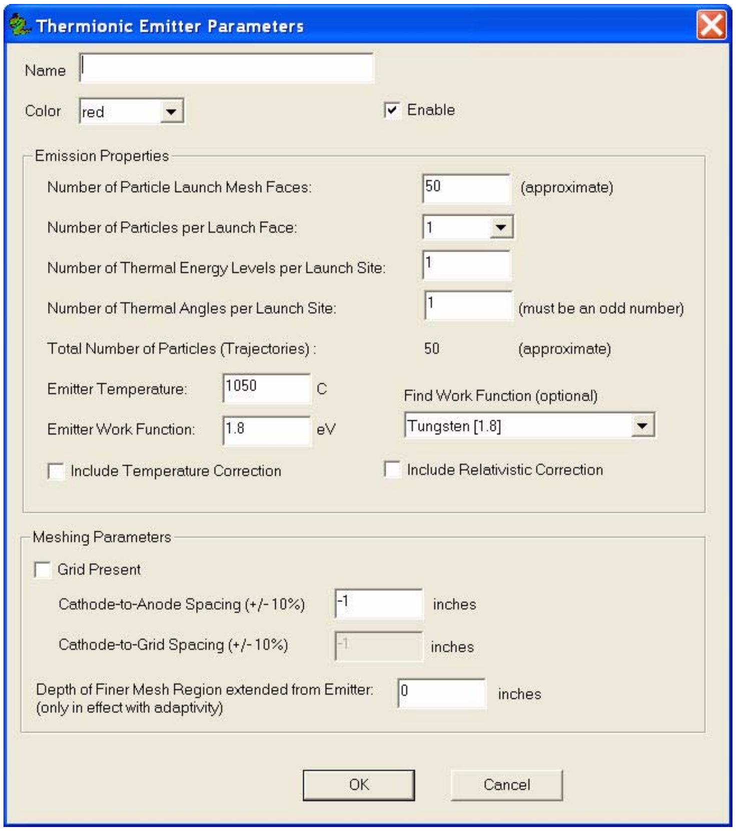
3.8.2.2 Emitter Assignments

The *Emitter Assignments* section allows you to add thermionic emitters, injected emitters and secondary emitters. The sections below describe the details for defining each emitter type.

Note that BOAGui provides the capability to disable particular emitters for a simulation without having to delete them from the case. This feature is useful if you want to temporarily turn off some secondary emitters or certain thermionic emitters for a simulation but want to re-use them in subsequent runs. By definition, a beam analysis requires at least one enabled primary (thermionic or injected) emitter. BoaGui will issue a warning if there is not at least one primary emitter specified and enabled.

Thermionic Emitters

To add a thermionic emitter, click on the *Thermionic Emitter Add* button beneath the emitter table. This will bring up the main window containing the geometric model. Find the desired surface and select it by pressing Ctrl-Left mouse button. Click Enter to complete the selection. The dialog shown in Figure 24 will appear.



Name:

Color: red ☑ Enable

Emission Properties

Number of Particle Launch Mesh Faces: 50 (approximate)

Number of Particles per Launch Face: 1

Number of Thermal Energy Levels per Launch Site: 1

Number of Thermal Angles per Launch Site: 1 (must be an odd number)

Total Number of Particles (Trajectories): 50 (approximate)

Emitter Temperature: 1050 C

Emitter Work Function: 1.8 eV

Find Work Function (optional): Tungsten [1.8]

☐ Include Temperature Correction ☐ Include Relativistic Correction

Meshing Parameters

☐ Grid Present

Cathode-to-Anode Spacing (+/- 10%) -1 inches

Cathode-to-Grid Spacing (+/- 10%) -1 inches

Depth of Finer Mesh Region extended from Emitter: (only in effect with adaptivity) 0 inches

FIGURE 24. REPLACE (new Tag filed) Thermionic Emitter Parameters dialog

Enter the desired *Name* and select a *Color.* (If the surface is already named in the geometry model, the *Name* will not be editable.) Use the *Enable* checkbox to enable or disable the emitter for the simulation. Disabling an emitter allows you to temporarily turn off an emitter without deleting it.

Enter the parameters in the *Emission Properties* box, including:

- *Number of Particle Launch Mesh Faces* - This value is an estimate only. The exact number will be calculated when the initial mesh is generated.
- Number of Particles per Launch Face

- *Number of Thermal Energy Levels per Launch Site*
- *Number of Thermal Angles per Launch Site* - This must be an odd number. BOA defines the angles such that one angle is always normal to the surface and the remaining angles form symmetrical pairs with respect to the normal.
- *Total Number of Particles (Trajectories)* - For convenience, the user interface displays the approximate *Total Number of Particles (Trajectories)*. This is calculated as the product of the *Number of Particle Launch Mesh Faces*, *Number of Particle per Launch Face*, *Number of Thermal Angles per Launch Site* and *Number of Thermal Energy Levels per Launch Site.* (E.g., if the *Number of Particle Launch Mesh Faces* = 50, *Number of Particle per Launch Face = 3, Number of Thermal Energy Levels per Launch Site* = 2 and *Number of Thermal Angles per Launch Site* = 3, then the user interface will show *Approximate Total Number of Emitted Particles* = 900.) Keep in mind that since the number of launch mesh faces you enter is an approximation, this calculation is also an approximation. The exact number of emitted particles will be calculated when the initial mesh is generated.
- *Emitter Temperature* - Enter in degrees C.
- *Emitter Work Function* - Enter in electron Volts. Enter the value directly or by selecting a work function from the *Find Work Function (optional)* drop-down box. This drop-down box contains a ‘library’ of work functions for commonly used materials. You can add materials to the library by selecting ‘Add Your Own Material’ in the drop-down box and entering the desired values in the *Material Properties* dialog. (You can also access the *Material Properties* dialog via the *Tools->Material Properties* menu. See Section 3.12.3 for details.)
- Check the *Temperature Correction* checkbox to include a temperature correction factor. This optional correction is derived from Maxwellian energy distribution for low voltage region in the proximity of the thermionic emitting surfaces. The correction factor as a function of (Tc/V) is expressed as a series expansion given by Langmuir and et al.
- Check the *Relativistic Correction* checkbox to include a relativistic correction factor. This optional correction is for electron guns operating in relativistic regime. It is formulated from Child-Langmuir law and recasted into a series expansion by Lau and et al. as a function of (gamma - 1).

Enter the *Meshing Parameters* as follows:

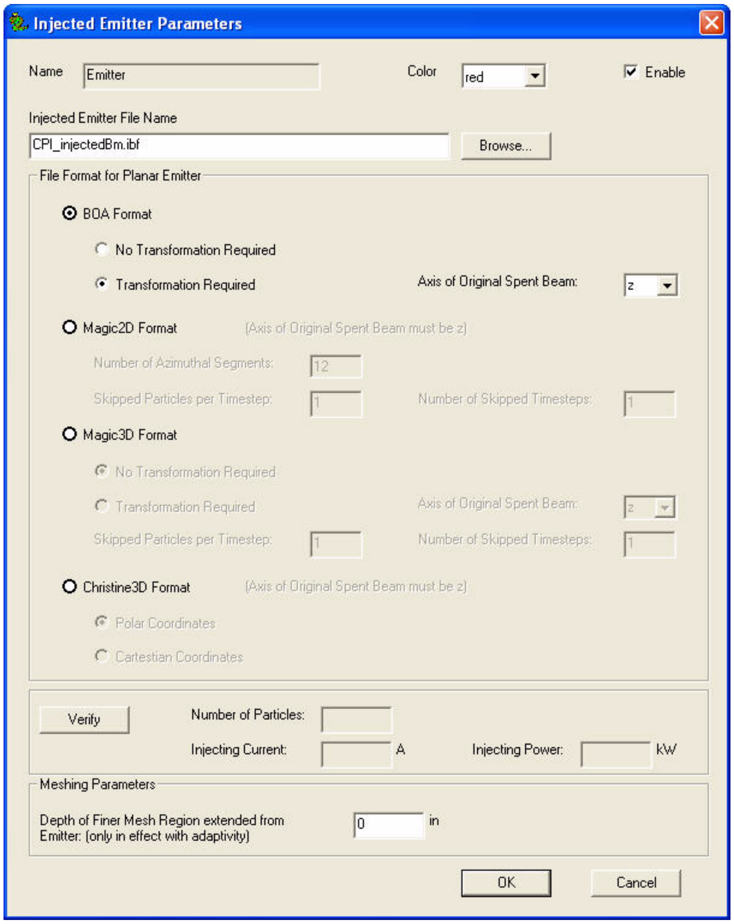
- The *Grid Present* checkbox determines how the cathode spacing is defined. If a grid is present, check the box and enter the value for *Cathode-to-Grid Spacing.* If no grid is present, leave the box unchecked and enter the value for the *Cathode-to-Anode Spacing*. The units for the spacing are the length units defined for the model (as defined in the geometry file). Note that the spacing value is approximate. It can vary +/-10%.
- The *Depth of Finer Mesh Region extended from Emitter* parameter allows you to define a region of finer mesh around the emitter. It is used only when adaptivity is turned on (i.e., when *Max Number of Adaptivity Passes* is set to > 0 on the *Execute BOA* dialog). The region is defined as a virtual column of user-specified depth extending from the emitter surface. The mesh within the column will be adaptively refined to approximately the same mesh size as on the emitter surface.

Click *OK* to complete the emitter definition. The emitter will appear in the table.

Injected Emitters

To add an injected emitter, click on the *Injected Emitter Add* button beneath the emitter table on the *Beam Optics* tab (see Figure 23). This will bring up the graphics display window containing the geometric model. Find the surface you want to define as the emitter and select it by pressing Ctrl-Left mouse button. Click enter to complete the selection. The dialog box shown in Figure 25 will appear.

Enter the desired *Name* for the injected emitter and select a *Color.* (If the surface is already named in the geometry model, the *Name* will not be editable.) Use the *Browse* button to locate the Injected Beam file. Use the *Enable* checkbox to enable or disable the emitter for the simulation. Disabling an emitter allows you to temporarily turn off an emitter without deleting it.



Name: Emitter
Color: red
☑ Enable
Injected Emitter File Name: CPI_injectedBm.ibf

File Format for Planar Emitter

● BOA Format
○ No Transformation Required
● Transformation Required
Axis of Original Spent Beam: z

○ Magic2D Format (Axis of Original Spent Beam must be z)
Number of Azimuthal Segments: 12
Skipped Particles per Timestep: 1
Number of Skipped Timesteps: 1

○ Magic3D Format
● No Transformation Required
○ Transformation Required
Axis of Original Spent Beam: z
Skipped Particles per Timestep: 1
Number of Skipped Timesteps: 1

○ Christine3D Format (Axis of Original Spent Beam must be z)
● Polar Coordinates
○ Cartesian Coordinates

Number of Particles:
Injecting Current: A
Injecting Power: kW

Meshing Parameters

Depth of Finer Mesh Region extended from Emitter: (only in effect with adaptivity) 0 in

FIGURE 25. REPLACE (new Tag filed) Injected Emitter Parameters dialog

Specify the file format using the radio buttons in the *File Format for Planar Emitter* section. BOA supports four file formats:

- BOA format
- Magic2D format
- Magic3D format
- Christine 3D format

The selections associated with each format are described below. The file formats are defined in the Appendix.

- **BOA Format** - The *BOA* file format is for a DC, Cartesian beam. It contains the 3D particle positions and velocities at the original injected surface/plane. If the specified emitter surface is a flat plane having a regular shape such as a circular disk or a rectangular plate, two options will be available:*No Transformation Required* and *Transformation Required.* Select *No Transformation Required* if the beam, as defined in the file, can be used exactly as is (i.e., if the initial particle positions in the file match the surface you designated as the injected emitter). If the two planes do not match, select *Transformation Required* and specify the *Axis of Original Spent Beam*. The original injected plane will be transformed to the actual injected plane.

If the specified emitter surface is not a flat plane having a regular shape, only the *No Transformation Required* option will be available. The specified emitter surface must approximate the initial particle positions in the file.

- **Magic 2D Format** - The *Magic2D* file format is for a Polar beam. *Number of Azimuthal Sections* is the number of azimuthal coordinates that Boa uses to convert a Magic2D spent beam (which is an axisymmetric beam) to a 3D Cartesian beam. *Skipped Particles per Timestep* is the number of particles to be skipped per time step when reading the Magic2D/3D spent beam data to reduce computation costs. Enter ‘1’ to use all particles. *Number of Skipped Timesteps* is the number of time steps to be skipped when reading the Magic2D/3D spent beam data file to reduce computational costs. Enter ‘1’ to use all steps.

  This format is available only if the specified emitter surface is a flat plane having a regular shape such as a circular disk or a rectangular plate. The axis of the original spent beam (i.e., the spent beam defined in the file) must be the z axis. Thus, the original injected plane will always be the r plane normal to the z-axis, and the direction of the beam is assumed to always point to the direction of the positive z-axis. The original injected plane will be transformed to the actual injected plane.

- **Magic 3D Format** - The *Magic3D* file format is for a DC, Cartesian beam. As with the BOA format, it contains the complete 3D particle attributes at the original injected surface. If the specified emitter surface is a flat plane having a regular shape such as a circular disk or a rectangular plate, two options will be available: *No Transformation Required* and *Transformation Required*. Select *No Transformation Required* if the beam, as defined in the file, can be used exactly as is (i.e., if the initial particle positions in the file match the surface you designated as the injected emitter). If the two planes do not match, select *Transformation Required* and specify the *Axis of Original Spent Beam*. The original injected plane will be transformed to the actual injected plane.

  If the specified emitter surface is not a flat plane having a regular shape, only the *No Transformation Required* option will be available. The specified emitter surface must approximate the initial particle positions in the file.

  Skipped Particles per Timestep is the number of particles to be skipped per time step when reading the Magic2D/3D spent beam data to reduce computation costs. Enter ‘1’ to use all particles. *Number of Skipped Timesteps* is the number of time steps to be skipped when reading the Magic2D/3D spent beam data file to reduce computational costs. Enter ‘1’ to use all steps.

- **Christine Format** - The *Christine* file format is available only if the specified emitter surface is a flat plane having a regular shape such as a circular disk or a rectangular plate. Specify either *Polar Coordinates* or *Cartesian Coordinates*. The axis of the original spent beam (the spent beam defined in the file) is assumed to be the z-axis. Thus, the original injected plane will always be the xy or r plane normal to the z-axis, and the direction of the beam is assumed to point in the direction of the positive z-axis. The original injected plane will be transformed to the actual injected plane.

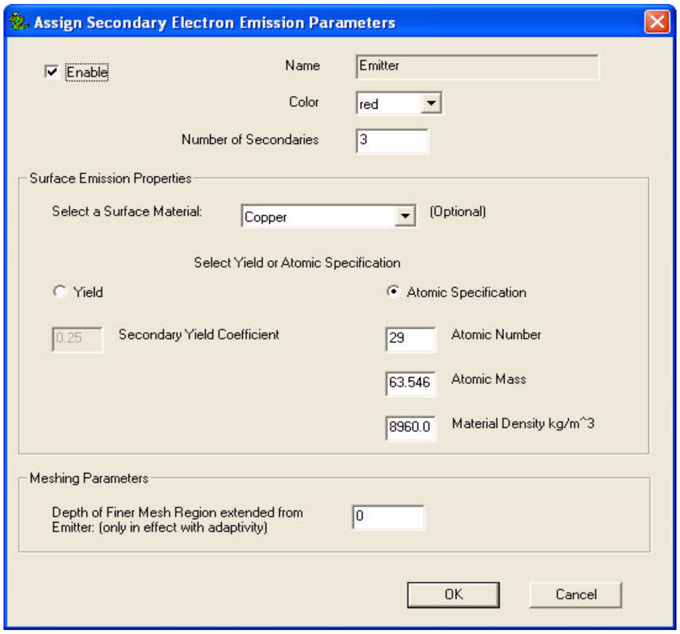
The second section of the *Injected Emitter Parameters* dialog allows you to verify that the values for the *Number of Particles*, the *Injecting Current* and the *Injecting Power* are what you expect. Click the *Verify* button to perform the verification.

In the *Meshing Parameters* section, the *Depth of Finer Mesh Region extended from Emitter* parameter allows you to define a region of finer mesh around the emitter. It is used only when adaptivity is turned on (i.e., when *Max Number of Adaptivity Passes* is set to > 0 on the *Execute BOA* dialog). The region is defined as a virtual column of user-specified depth extending from the emitter surface. The mesh within the column will be adaptively refined to approximately the same mesh size as on the emitter surface.

Click *OK* to complete the injected emitter definition. The injected emitter will appear in the *Emitter Assignments* table on the *Beam Optics* tab of the *Attribute Assignment Properties* dialog.

Secondary Emitters

To add a secondary emitter, click on the *Secondary Emitter Add* button beneath the emitter table on the *Beam Optics* tab (see Figure 23). This will bring up the graphics display window containing the geometric model. Find the surface you want to define as a secondary emitter and select it by pressing Ctrl-Left mouse button. Click enter to complete the selection. The dialog box shown in Figure 26 will appear.



Assign Secondary Electron Emission Parameters

☑ Enable

Name: Emitter

Color: red

Number of Secondaries: 3

Surface Emission Properties

Select a Surface Material: Copper (Optional)

Select Yield or Atomic Specification

○ Yield

● Atomic Specification

Secondary Yield Coefficient: 0.25

Atomic Number: 29

Atomic Mass: 63.546

Material Density kg/m^3: 8960.0

Meshing Parameters

Depth of Finer Mesh Region extended from Emitter: (only in effect with adaptivity): 0

FIGURE 26. REPLACE (new Tag filed) Secondary

Enter the desired *Name* and select a *Color*. (If the surface is already named in the geometry model, the *Name* will not be editable.) In the *Number of Secondaries* field, enter the number of scattered electrons caused by one incoming particle. Use the *Enable* checkbox to enable or disable the emitter for the simulation. (Disabling an emitter allows you to temporarily turn off an emitter without deleting it)

In the *Surface Emission Properties* section, use the radio buttons to select the desired secondary model. The *Yield* option uses a simple *Secondary Yield Coefficient* to compute the yield. The *Atomic Specification* option uses a Monte Carlo simulation to compute the yield based on the values entered for *Atomic Number*, *Atomic Mass* and *Material Density* (in grams per cubic centimeter).

For either secondary model, you can enter the required parameters in one of two ways:

- Enter the parameter values directly, or

- Select a material from the *Select a Surface Material* drop-down box, located above the radio buttons. BOAGui will automatically populate the values associated with the selected material, in the *Material Properties* database for each parameter. (See Section 3.12.3 for a discussion of the *Material Properties* database.)

The *Number of Secondaries* is used with the *Maximum Number of Secondary Generations* (defined on the *Beam Optics* tab of the *Assign Attributes* dialog) as shown in the following example:

> Assume *Number of Secondaries* = 3 and *Maximum No. of Secondary Generations* = 2. When a primary hits a secondary emitter, three electrons are emitted. When these three secondaries of the 1st generation hit another secondary emitter, another three electrons of the 2nd generation are emitted. When these 2nd generation secondaries hit another secondary emitter they do not propagate any further, since there is a maximum of two generations.

Click *OK* to complete the emitter definition. The emitter will appear in the table.

To modify an emitter once it is defined, select the emitter in the emitter table and click the *Modify* button. The appropriate dialog will reappear and you can edit the values. To delete an emitter, select the emitter in the emitter table and click the *Delete* button.

3.8.2.3 External Magnetic Field Options

The *External Magnetic Field Options* section on the *Beam Optics* tab (refer to Figure 23) enables you to define an external magnetic field. It is active when the *Analysis* menu is set to either ‘Electro/Magnetostatic Beam’ or ‘Electro/Magneto/Self Field Beam’. The drop down box contains the following options:

1. Uniform Magnetic Flux Density
2. Boa Integrated Magnetic Case
3. Boa Magnetic Flux Density (fef file)
4. CCR Imported Magnetic Flux Density (bf file)
5. Maxwell 3D Imported Magnetic Flux Density
6. Maxwell 2D XY Imported Magnetic Flux Density
7. Maxwell 2D RZ Imported Magnetic Flux Density
8. Pandira/Poisson RZ Imported Magnetic Flux Density
9. 1D Axial Imported Magnetic Flux Density

Choose the desired option and a dialog will appear prompting you to enter the necessary parameters. The specifics of the dialog will vary depending on the selected *External Magnetic Field Option* but, in general, all of the dialogs are similar and contain the same basic elements:

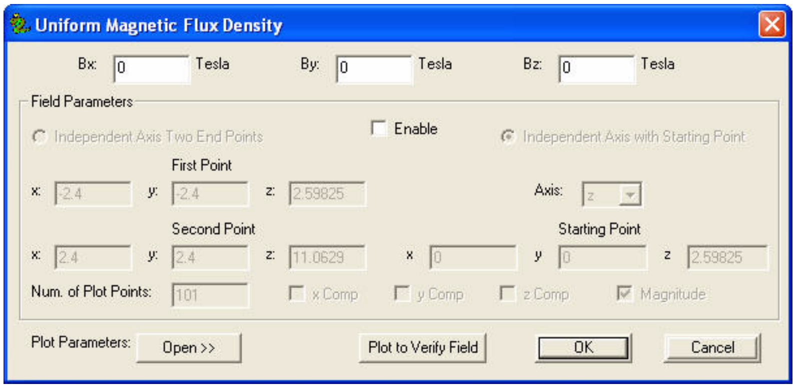
- Field Definition - The upper portion of the dialogs includes all parameters necessary to define the selected magnetic field type in three dimensions.

- Field Plot - The *Plot to Verify Field* button allows you to create a linear plot of the magnetic flux density in order to verify that the field is defined correctly.
  - The *Field Parameters* section provides options for plotting the field data. To make this section active, check the Enable checkbox. *{TBD, expand this section to include discussion of all fields}*
  - The *Plot Parameters* opens a hidden section of the dialog that provides additional controls such as *Plot Characteristics* (i.e., the appearance of the plot, including title, axis labels, etc.) *{TBD: Refer to the Plot section earlier in the manual once that is complete.}*

The sections below describe the dialogs triggered by each *External Magnetic Field Option*:

Uniform Magnetic Flux Density

Setting the *External Magnetic Field Options* dropdown box to ‘Uniform Magnetic Flux Density’ allows direct entry of a uniform magnetic flux density. This option brings up the dialog shown in Figure 27. Enter the three components of the uniform magnetic flux density (*Bx*, *By*, *Bz*) in Tesla.



Bx: 0 Tesla By: 0 Tesla Bz: 0 Tesla

☐ Enable

○ Independent Axis Two End Points ● Independent Axis with Starting Point

First Point x: -2.4 y: -2.4 z: 2.59825

Axis: z

Second Point x: 2.4 y: 2.4 z: 11.0629

Starting Point x 0 y 0 z 2.59825

Num. of Plot Points: 101 ☐ x Comp ☐ y Comp ☐ z Comp ☑ Magnitude

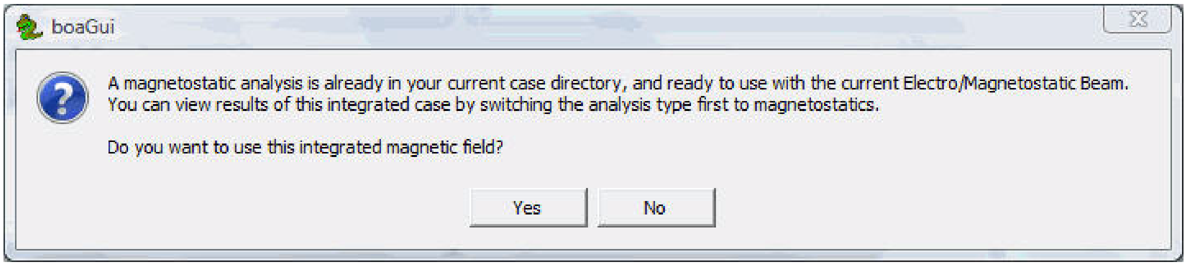
FIGURE 27. Dialog for defining a Uniform Magnetic Flux Density

Boa Integrated Magnetic Case

Setting the *External Magnetic Field Options* dropdown box to 'Boa Integrated Magnetic Case' implies that you want to use the magnetic fields, which have been generated by BOA from the same geometry of the currently open case. To use this option, the following should be true:

- the currently open case was initiated as a Magnetostatic case (*Analysis* menu item set to 'Magnetostatic'),
- you have run the mesh for and executed that Magnetostatic case (so that the magnetic field exists), and
- you have now changed the *Analysis* type of the current case to either 'Electro/Magnetostatic Beam' or to 'Electro/Magneto/Self Field Beam').

If you create such a case in this way, then setting the *External Magnetic Field Options* dropdown box to ‘Boa Integrated Magnetic Case’ will bring up the message in Figure 28. Click ‘Yes’ to complete your *External Magnetic Field* selection.

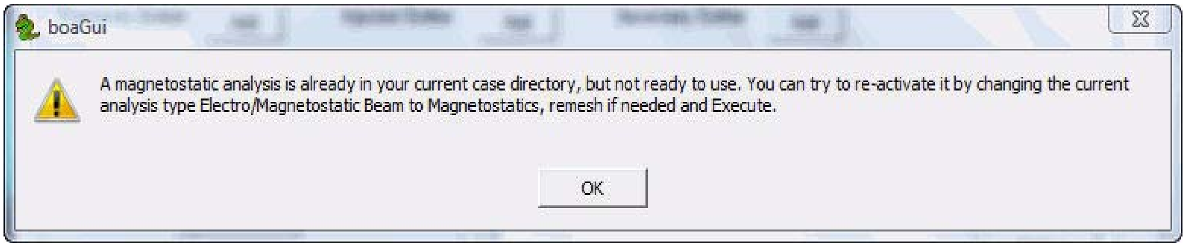


A magnetostatic analysis is already in your current case directory, and ready to use with the current Electro/Magnetostatic Beam. You can view results of this integrated case by switching the analysis type first to magnetostatics.

Do you want to use this integrated magnetic field?

FIGURE 28. A Magnetostatic analysis exists in current case and is ready to use

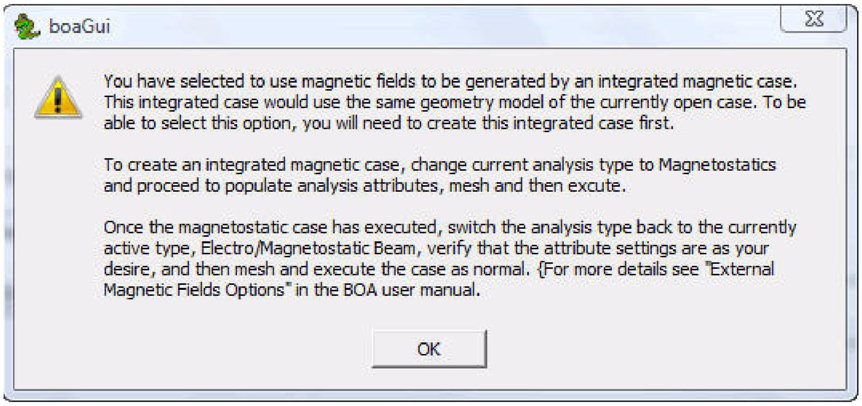
If a magnetostatic case exists in the current directory but is incomplete (i.e., the case started as a ‘Magnetostatic’ case but the case was never executed), then setting the *External Magnetic Field Options* dropdown box to ‘Boa Integrated Magnetic Case’ will bring up the informational message shown in Figure 29. This message indicates that you must go back and complete the Magnetostatic analysis before continuing with the integrated case. (That is, first change the *Analysis* type back to ‘Magnetostatic, mesh (is needed) and execute the case.)



A magnetostatic analysis is already in your current case directory, but not ready to use. You can try to re-activate it by changing the current analysis type Electro/Magnetostatic Beam to Magnetostatics, remesh if needed and Execute.

FIGURE 29. A Magnetostatic analysis exists in current case but is not ready to use

If there is no magnetostatic case in the current directory (i.e., the case was simply started as an ‘Electro/Magnetostatic Beam’ or ‘Electro/Magneto/Self Field Beam’ analysis), then setting the *External Magnetic Field Options* dropdown box to ‘Boa Integrated Magnetic Case’ will bring up the informational message shown in Figure 30. This message explains how to create an integrated magnetostatic case.



You have selected to use magnetic fields to be generated by an integrated magnetic case. This integrated case would use the same geometry model of the currently open case. To be able to select this option, you will need to create this integrated case first.

To create an integrated magnetic case, change current analysis type to Magnetostatics and proceed to populate analysis attributes, mesh and then excute.

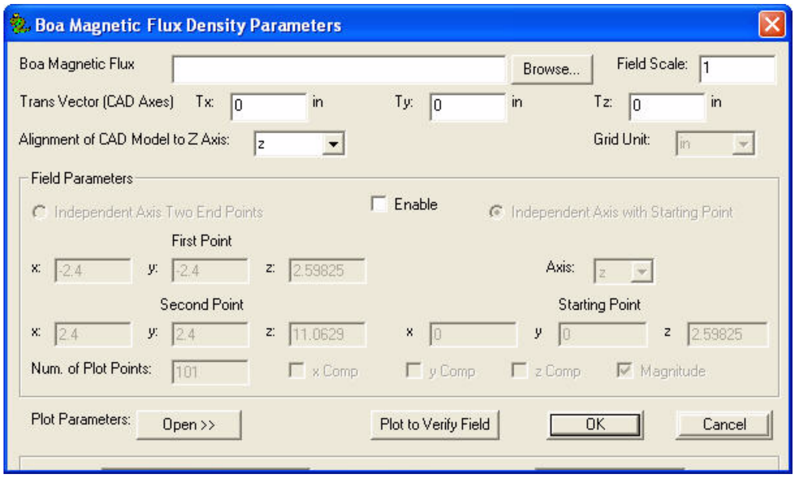
Once the magnetostatic case has executed, switch the analysis type back to the currently active type, Electro/Magnetostatic Beam, verify that the attribute settings are as your desire, and then mesh and execute the case as normal. {For more details see "External Magnetic Fields Options" in the BOA user manual.

FIGURE 30. A Magnetostatic analysis does not exist in the current case

Boa Magnetic Flux Density (fef file)

Setting the *External Magnetic Field Options* dropdown box to ‘Boa Magnetic Flux Density (fef file)’ enables you to use the magnetic flux density file generated by a previous BOA run. This option brings up the dialog box shown in Figure 31.

Use the *Browse* button to locate the desired *.fef file. The *.fef format is a CCR proprietary format and it is defined in the Appendix. The remaining fields are defined as follows:



Boa Magnetic Flux
Field Scale: 1
Trans Vector (CAD Axes) Tx: 0 in Ty: 0 in Tz: 0 in
Alignment of CAD Model to Z Axis: z
Grid Unit: in
○ Independent Axis Two End Points
☐ Enable
● Independent Axis with Starting Point
First Point x: -2.4 y: -2.4 z: 2.59825
Axis: z
Second Point x: 2.4 y: 2.4 z: 11.0629
Starting Point x 0 y 0 z 2.59825
Num. of Plot Points: 101
☐ x Comp ☐ y Comp ☐ z Comp ☑ Magnitude

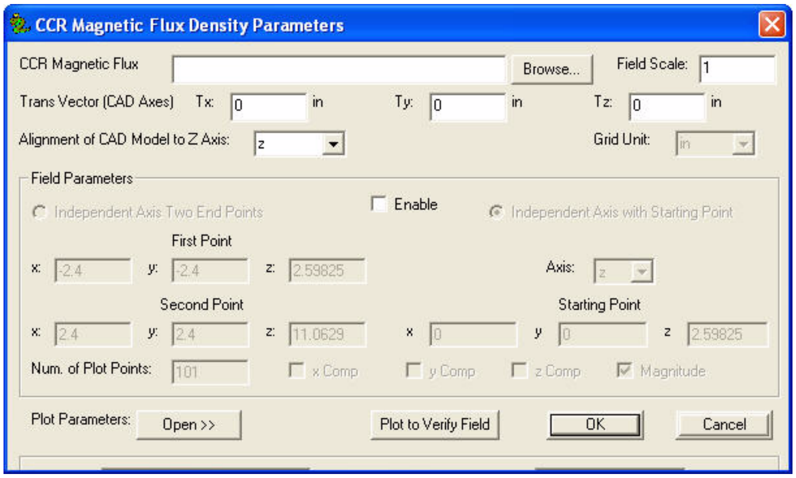
FIGURE 31. Dialog for importing a Magnetic Flux Density File in *Boa* format

- The *Field Scale* is the scale factor for the imported magnetic field magnitude. Set it to ‘1’ to use the imported field as is.
- The three components of the *Translation Vector (CAD Axes)* are required if the origin of the imported magnetic model is different from that of the geometry (CAD) model. The translation vector is used to match the two origins. For example, if the origin of the imported magnetic model is (2,1,5) and the geometry model is (3,0,0) then the translation vector is (3-2, 0-1, 0-5) = (1,-1,-5) where the elements of each tuple are the x, y and z coordinates respectively.
- If the z-axis of the geometry model is different from the z-axis of the imported magnetic model, a coordinate rotation is also required. Use the *Alignment of CAD Model to Z Axis* dropdown box to specify which coordinate axis of the CAD model (x, y or z) to match to the z-axis of the imported magnetic model.
- *Grid Unit* is the length unit of the imported field, which can be different from the model.

CCR Imported Magnetic Flux Density (bf file)

Setting the *External Magnetic Field Options* dropdown box to the ‘CCR Imported Magnetic Flux Density (bf file)’ option enables you to import a magnetic flux density file in the CCR format. This option brings up the dialog box shown in Figure 32.

Use the *Browse* button to locate the desired CCR format Magnetic Flux Density file (*.bf file).



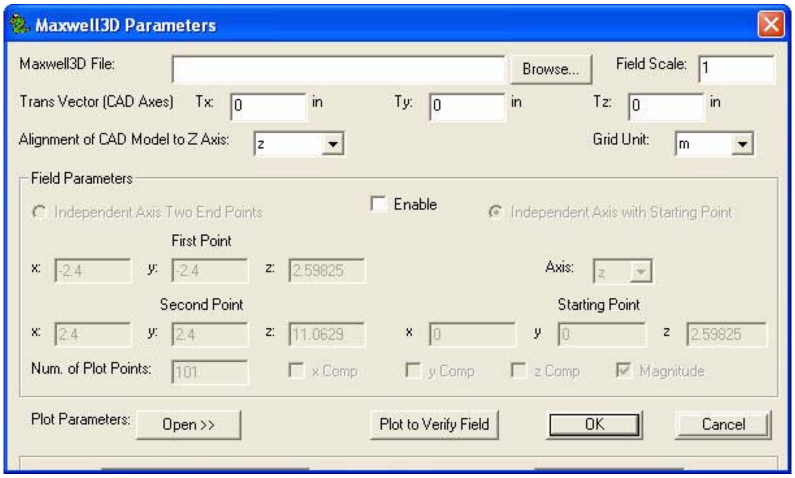
CCR Magnetic Flux
Field Scale: 1
Trans Vector (CAD Axes) Tx: 0 in Ty: 0 in Tz: 0 in
Alignment of CAD Model to Z Axis: z
Grid Unit: in
○ Independent Axis Two End Points
☐ Enable
● Independent Axis with Starting Point
First Point x: -2.4 y: -2.4 z: 2.59825
Axis: z
Second Point x: 2.4 y: 2.4 z: 11.0629
Starting Point x 0 y 0 z 2.59825
Num. of Plot Points: 101
☐ x Comp ☐ y Comp ☐ z Comp ☑ Magnitude

FIGURE 32. Dialog for importing a Magnetic Flux Density File in *CCR* format

The *.bf format is a CCR proprietary format and is defined in the Appendix. Specify the *Field Scale, Translation Vector (CAD Axes), Alignment of CAD Model to Z Axis* and *Grid Unit*, if appropriate. (See BOA Magnetic Flux Density (fef file) section above for definitions of these parameters.)

Maxwell 3D Imported Magnetic Flux Density

Setting the *External Magnetic Field Options* dropdown box to the ‘Maxwell 3D Imported Magnetic Flux Density’ option enables you to import a magnetic flux density file in the Maxwell 3D format. This option brings up the dialog box shown in Figure 33.



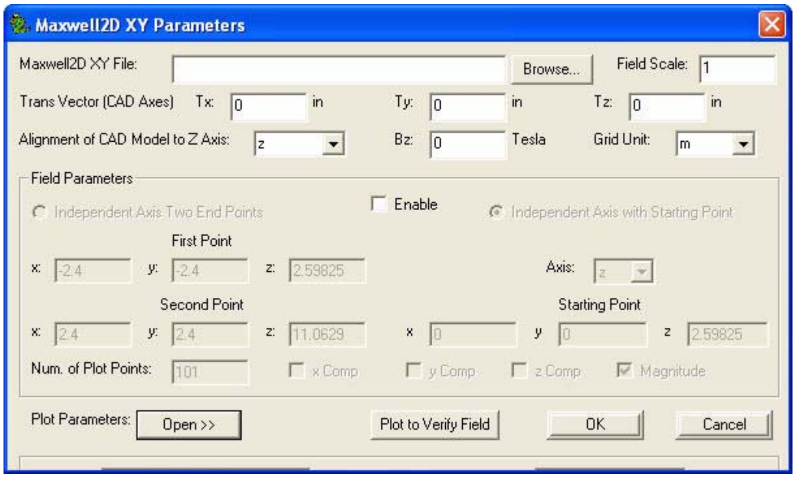
Maxwell3D File:
Field Scale: 1
Trans Vector (CAD Axes) Tx: 0 in Ty: 0 in Tz: 0 in
Alignment of CAD Model to Z Axis: z
Grid Unit: m
○ Independent Axis Two End Points ☐ Enable ● Independent Axis with Starting Point
First Point x: -2.4 y: -2.4 z: 2.59825
Axis: z
Second Point x: 2.4 y: 2.4 z: 11.0629
Starting Point x 0 y 0 z 2.59825
Num. of Plot Points: 101 ☐ x Comp ☐ y Comp ☐ z Comp ☑ Magnitude
Plot Parameters: Open >>

FIGURE 33. Dialog for importing a Magnetic Flux Density File in *Maxwell 3D* format

Use the *Browse* button to locate the desired Maxwell 3D file. Specify the *Field Scale*, *Translation Vector (CAD Axes), Alignment of CAD Model to Z Axis* and *Grid Unit*, if appropriate. (See *BOA Magnetic Flux Density (fef file)* section above for definitions of these parameters.)

Maxwell 2D XY Imported Magnetic Flux Density

Setting the *External Magnetic Field Options* dropdown box to the ‘Maxwell 2D XY Imported Magnetic Flux Density’ option allows you to import a magnetic field from Maxwell2D in XY coordinates. This option brings up the dialog box shown in Figure 34.



Maxwell2D XY File:
Field Scale: 1
Trans Vector (CAD Axes) Tx: 0 in Ty: 0 in Tz: 0 in
Alignment of CAD Model to Z Axis: z
Bz: 0 Tesla
Grid Unit: m
○ Independent Axis Two End Points ☐ Enable ● Independent Axis with Starting Point
First Point x: -2.4 y: -2.4 z: 2.59825
Axis: z
Second Point x: 2.4 y: 2.4 z: 11.0629
Starting Point x 0 y 0 z 2.59825
Num. of Plot Points: 101 ☐ x Comp ☐ y Comp ☐ z Comp ☑ Magnitude
Plot Parameters: Open >>

FIGURE 34. Dialog for importing a Magnetic Flux Density File in a *Maxwell 2D, XY coordinate* format

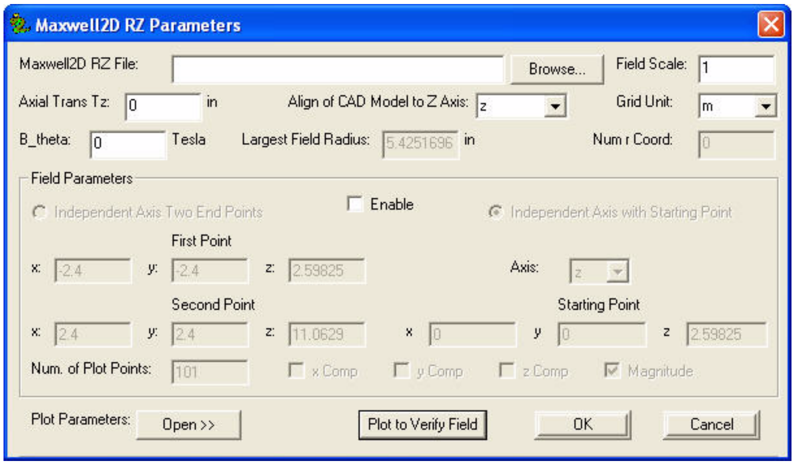
Use the *Browse* button to locate the desired Maxwell 2D file. Specify the *Field Scale*, *Translation Vector (CAD Axes), Alignment of CAD Model to Z Axis* and *Grid Unit*, if appropriate. (See *BOA Magnetic Flux Density (fef file)* section above for definitions of these parameters.)

Use the *Bz* edit box to specify the uniform flux component (i.e., the z component of the magnetic flux) in Tesla.

Maxwell 2D RZ Imported Magnetic Flux Density

Setting the *External Magnetic Field Options* dropdown box to the ‘Maxwell 2D RZ Imported Magnetic Flux Density’ option allows you to import a magnetic field from Maxwell2D in RZ coordinates. This option brings up the dialog box shown in Figure 35.

Use the *Browse* button to locate the desired Maxwell 2D file. The remaining fields are defined as follows:



Maxwell2D RZ File:
Field Scale: 1
Axial Trans Tz: 0 in
Align of CAD Model to Z Axis: z
Grid Unit: m
B_theta: 0 Tesla
Largest Field Radius: 5.4251696 in
Num r Coord: 0
○ Independent Axis Two End Points
☐ Enable
● Independent Axis with Starting Point
First Point x: -2.4 y: -2.4 z: 2.59825
Axis: z
Second Point x: 2.4 y: 2.4 z: 11.0629
Starting Point x 0 y 0 z 2.59825
Num. of Plot Points: 101
☐ x Comp ☐ y Comp ☐ z Comp ☑ Magnitude

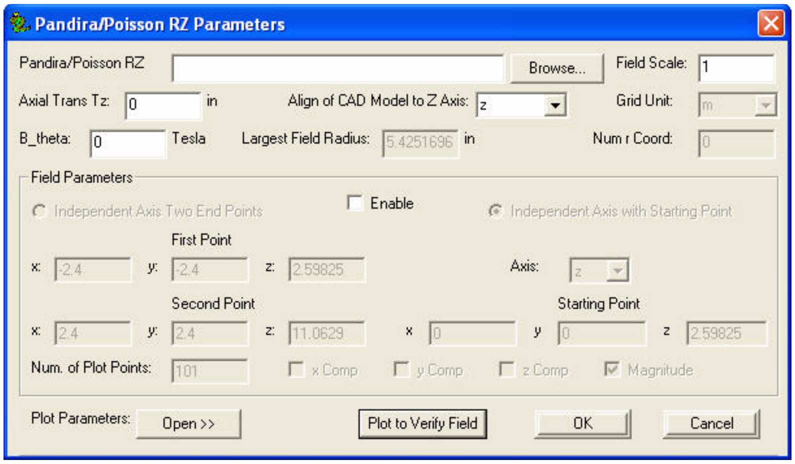
FIGURE 35. Dialog for importing a Magnetic Flux Density File in a *Maxwell 2D, RZ coordinate* format

- The *Field Scale* is the scale factor for the imported magnetic field magnitude. Set it to ‘1’ to use the imported field as is.
- The *Axial Translation (Tz)* is used to match the z-coordinate of the origin of the imported flux to that of the corresponding axis of the CAD model (the beam axis).
- If the z-axis of the geometry model is different from the z-axis of the imported magnetic model, a coordinate rotation is also required. Use the *Alignment of CAD Model to Z* dropdown box to specify which coordinate axis of the CAD model (x, y or z) to match to the z-axis of the imported magnetic model.
- B_{theta} defines the 3rd uniform flux component (theta component of the magnetic flux) in Tesla.
- The *Largest Field Radius* defines how far from the axis to compute B_r
- The *Number of r Coordinates of the Field* specifies how many lines to compute

Pandira RZ Imported Magnetic Flux Density

Setting the *External Magnetic Field Options* dropdown box to the ‘Pandira RZ Imported Magnetic Flux Density’ option allows you to import a magnetic field from Pandira in RZ coordinates. This option brings up the dialog box shown in Figure 36.

Use the *Browse* button to locate the desired Pandira file. The remaining parameters in the dialog box are identical to those for the *Maxwell 2D RZ Magnetic Flux Density* option. See the previous section for details.


Pandira/Poisson RZ Parameters
Browse...
Field Parameters
First Point
Second Point
Starting Point
Plot Parameters:
Open >>
Plot to Verify Field
OK
Cancel

Pandira/Poisson RZ:
Field Scale: 1
Axial Trans Tz: 0 in
Align of CAD Model to Z Axis: z
Grid Unit: m
B_theta: 0 Tesla
Largest Field Radius: 5.4251696 in
Num r Coord: 0
○ Independent Axis Two End Points
☐ Enable
● Independent Axis with Starting Point
First Point x: -2.4 y: -2.4 z: 2.59825
Axis: z
Second Point x: 2.4 y: 2.4 z: 11.0629
Starting Point x 0 y 0 z 2.59825
Num. of Plot Points: 101
☐ x Comp ☐ y Comp ☐ z Comp ☑ Magnitude

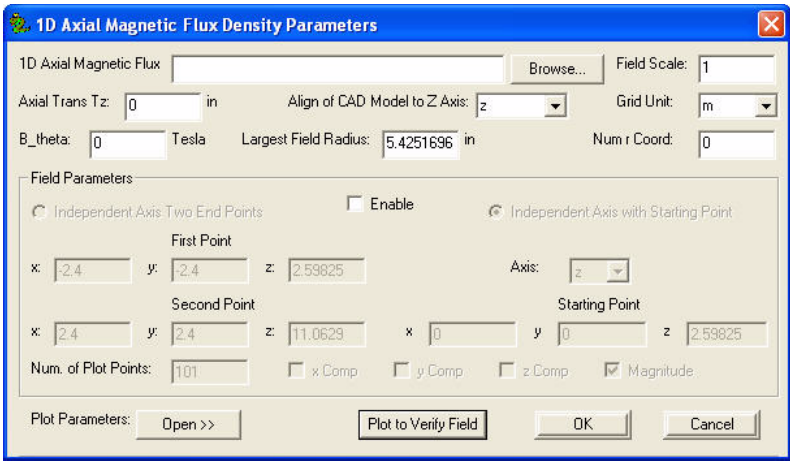
FIGURE 36. Dialog for importing a Magnetic Flux Density File in a *Pandira, RZ coordinate* format

1D Axial Imported Magnetic Flux Density

Setting the *External Magnetic Field Options* dropdown box to the ‘1D Axial Imported Magnetic Flux Density’ option allows you to import a 1D axial magnetic flux density file. This option brings up the dialog box shown in Figure 37.

Use the *Browse* button to locate the desired file. The format for the 1D Axial file is defined in the Appendix.

The remaining parameters in the dialog box are identical to those for the *Maxwell 2D RZ Magnetic Flux Density* option. See the previous section for details. Note that BOA calculates the off-axis or 2nd component of the flux (B_r) internally.

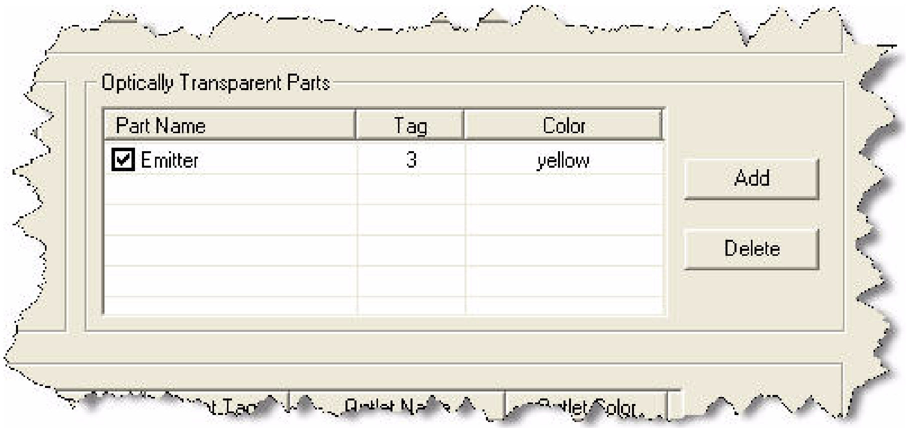

1D Axial Magnetic Flux Density Parameters
Browse...
Field Parameters
First Point
Second Point
Starting Point
Plot Parameters:
Open >>
Plot to Verify Field
OK
Cancel

1D Axial Magnetic Flux:
Field Scale: 1
Axial Trans Tz: 0 in
Align of CAD Model to Z Axis: z
Grid Unit: m
B_theta: 0 Tesla
Largest Field Radius: 5.4251696 in
Num r Coord: 0
○ Independent Axis Two End Points
☐ Enable
● Independent Axis with Starting Point
First Point x: -2.4 y: -2.4 z: 2.59825
Axis: z
Second Point x: 2.4 y: 2.4 z: 11.0629
Starting Point x 0 y 0 z 2.59825
Num. of Plot Points: 101
☐ x Comp ☐ y Comp ☐ z Comp ☑ Magnitude

FIGURE 37. Dialog for importing a Magnetic Flux Density File in *1D Axial* format

3.8.2.4 Optically Transparent Parts

The *Optically Transparent Parts* section of the *Attribute Assignment* dialog *Beam Optics* tab enables you to define parts in the model as optically transparent.



Part Name	Tag	Color
☑ Emitter	3	yellow

FIGURE 38. *Optically Transparent Parts* table (on the *Beam Optics* tab of the *Attribute Assignments Properties dialog*)

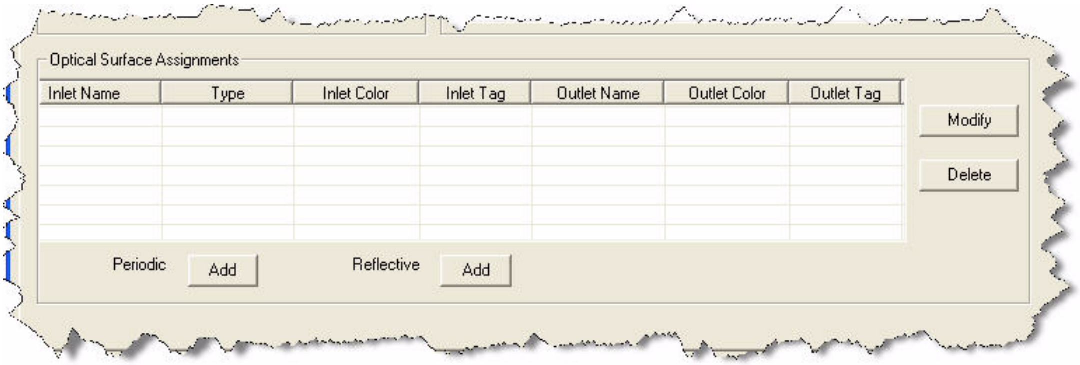
To make a part optically transparent, click the *Add* button next to the table shown in Figure 38. (Refer back to Figure 23 to see the complete *Beam Optics* tab.) This will bring up the graphics display window containing the geometric model. Find the desired part and select it by pressing Ctrl-Left mouse button. Click enter to select. The part will appear in the table.

To delete optical transparency from a part, highlight the part in the table and click the *Delete* button.

Optical transparency for a part can be enabled or disabled via the checkbox that precedes the part name in the *Optically Transparent Parts* table (see Figure 38). When the checkbox is checked, the part is enabled.

3.8.2.5 Optical Surface Assignments

The *Optical Surface Assignments* section of the *Beam Optics* tab (Figure 39) is used to define *Periodic Surfaces* and *Reflective Surfaces* in order to reduce the size of the model. (See Figure 23 for the complete *Beam Optics* tab of the *Attribute Assignments Properties dialog*)

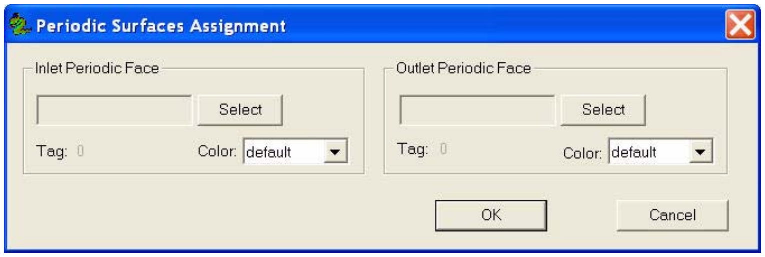


Inlet Name	Type	Inlet Color	Inlet Tag	Outlet Name	Outlet Color	Outlet Tag

FIGURE 39. *Optical Surface Assignments* table (on the *Beam Optics* tab of the *Attribute Assignments Properties dialog*)

Periodic Surfaces

Periodic surfaces are used to model very long parts. Periodic surface assignment entails specifying two surfaces: an inlet surface and an outlet surface. When particles exit the outlet surface, particles with identical attributes (e.g., velocity, current) re-enter the inlet surface. In effect, it is as if the model is infinitely repeating itself.



Tag: 0
Color: default

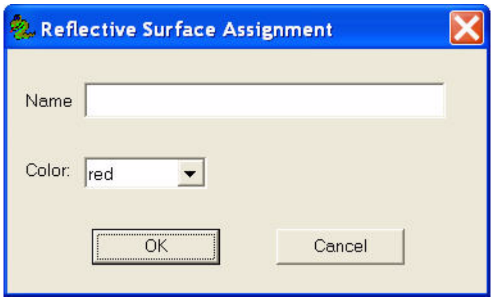
Tag: 0
Color: default

FIGURE 40. Periodic Surfaces Assignment dialog

To add a pair of periodic surfaces, click on the *Periodic Add* button beneath the *Optical Surface Assignments* table (shown in Figure 39). This will bring up the *Periodic Surfaces Assignment* dialog shown in Figure 40. To define the inlet periodic surface, click the *Select* button in the *Inlet Periodic Face* section. This will bring up the window containing the geometric model. Find the surface you want to define as the inlet periodic face and select it by pressing Ctrl-Left mouse button. Click enter to complete the selection. The *Periodic Surfaces Assignment* dialog will reappear. Enter the desired *Name* (if editable) and select a *Color.* (Remember, the *Name* field will not be editable for any surface that already has a name in the CAD model.) To define the outlet periodic face, repeat the same process using the *Select* button and *Color* dropdown box in the *Outlet Periodic Surface* section. Click *OK* to complete the assignment. The periodic surfaces will appear as an entry in the table.

Reflective Surfaces

Reflective surfaces are used to model symmetry. During the simulation, when particles encounter reflective surfaces, they reflect back like a ray of light hitting a mirror. So, for example, instead of modeling a complete sphere, you could model one octant of the sphere (a symmetric 1/8 ‘slice’) and then define the three ‘sliced’ faces as reflective. Similarly, for a Pierce gun, only 1/2 or 1/4 of the gun would be required.



Color: red

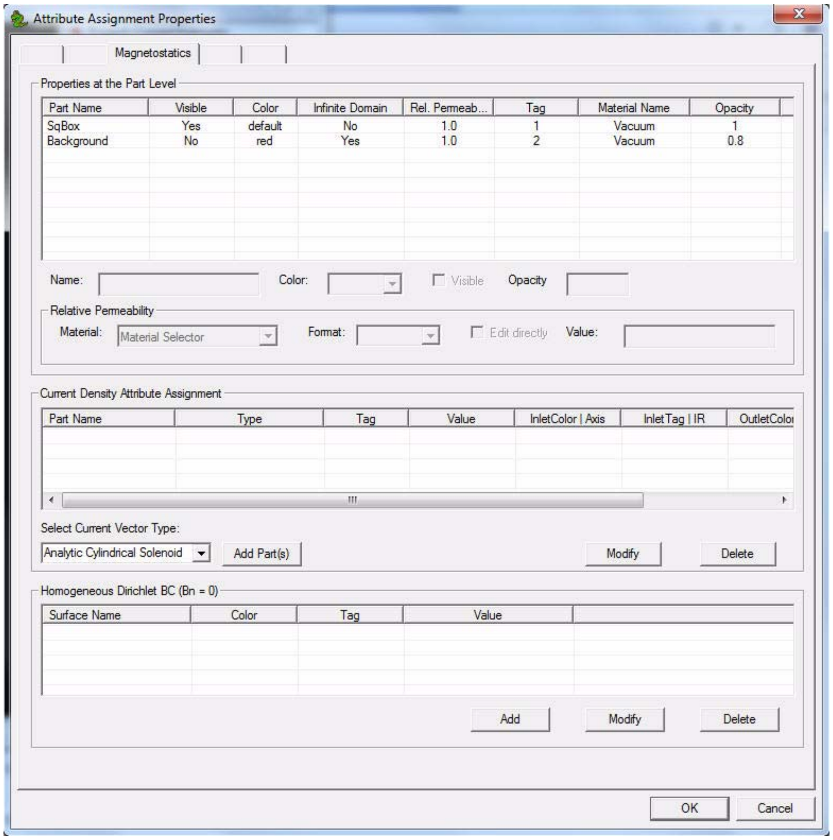
FIGURE 41. Reflective Surface Assignment dialog

To add a reflective surface, click on the *Reflective Add* button beneath the *Optical Surface Assignments* table (shown in Figure 39). This will bring up the containing the geometric model. Find the surface you want to define as reflective and select it by pressing Ctrl-Left mouse button. Click enter to select. The dialog shown in Figure 41 will appear. Enter the desired *Name* (if editable) and select a *Color.* (Remember, the *Name* field will not be editable for any surface that already has a name in the CAD model.) Click *OK* to complete the assignment. The reflective surface will appear as an entry in the table.

To modify an Optical Surfaces entry once it is defined, select it in the table and click the *Modify* button. The appropriate dialog will reappear and you can edit the values. To delete an entry, select the entry in the table and click the *Delete* button.

3.8.3 Magnetostatics Attributes

The *Magnetostatics* tab, shown in Figure 42 is enabled whenever the *Analysis* menu is set to the ‘Magnetostatic’ analysis type. Use it to make the attribute assignments required for magnetostatic analyses. It contains three sections. The top section (*Properties at the Part Level*) specifies the basic magnetostatic properties for all parts in the model. The middle section (*Current Density Attribute Assignment*) specifies current densities for parts that are conductors. The bottom section specifies homogenous Dirichlet boundary conditions for surfaces.


Attribute Assignment Properties
Magnetostatics
Properties at the Part Level
Name:
Color:
☐ Visible
Opacity
Relative Permeability
Material:
Material Selector
Format:
☐ Edit directly
Value:
Current Density Attribute Assignment
Add Part(s)
Modify
Delete
Homogeneous Dirichlet BC (Bn = 0)
Add
Modify
Delete
OK
Cancel

Part Name	Visible	Color	Infinite Domain	Rel. Permeab...	Tag	Material Name	Opacity
SqBox	Yes	default	No	1.0	1	Vacuum	1
Background	No	red	Yes	1.0	2	Vacuum	0.8

Part Name	Type	Tag	Value	InletColor | Axis	InletTag | IR	OutletColoı

Select Current Vector Type: Analytic Cylindrical Solenoid

Surface Name	Color	Tag	Value

FIGURE 42. Attribute Assignment dialog box, Magnetostatics tab

3.8.3.1 Properties at the Part Level

The *Properties at the Part Level* section enables you to assign attributes to the parts that make up the geometric model. The table at the top of the section summarizes all of the parts in the geometry file. To modify a part’s properties, first select the part to make it active. Properties include:

Name - The part name is identical to the name provided in the geometry file. This property is not editable.

Color dropdown box - Specify a color for the part.

Visible checkbox - Check/uncheck the box to show/hide the part in the graphics display region. BoaGui will initially set all parts to visible except for the background part.

Opacity - *Opacity* is the translucency of a part. BoaGui uses it to draw the part in the graphics display window. It ranges from 1 to 0, going from no transmission of light through the part (opaque) to full light transmission (transparent). A part's *Opacity* is editable only if it is set to *Visible*. (Notice that although by default the background part is set to *invisible*, if you change it to *visible* then its *Opacity* defaults to 0.8. This default value allows you to see the internal parts.)

Relative Permeability section - The *Relative Permeability* for a part can be entered directly or looked up in the *Materials Properties* database. (See Section 3.12.3 for a description of the *Materials Properties* database.) Typically it is looked up, so by default the *Format* and *Value* fields are non-editable (i.e., the *Edit directly* checkbox between them is unchecked) and the *Material* drop-down box is used to find the *Relative Permeability.* The fields in this section include:

> *Material* - This drop-down box contains a list of materials that correspond to those in the *Materials Properties* database. Select the desired material and the *Relative Permeability Value* will appear along with the *Format* in which it is entered.
>
> *Format* - There are three possibilities for the *Relative Permeability's Format*:
>
> - 'Scalar' - This type implies that the relative permeability is constant and uniform. That is, it is a single number. In the current version, most of the BOA provided materials in the *Material Properties* database have scalar relative permeabilities.
> - 'File' - This type implies that the relative permeability is provided in a two-column data file containing the BH curve. In the current version, the BOA provided *Material Properties* database contains three materials whose *Relative Permeability* is provided in file format.
> - 'Formula' type [this feature is not yet functional] - Future versions of BOA will allow BH curves to be entered via equations (eg B = 0.2 * H^0.2).
>
> *Edit Directly* checkbox - The *Edit Directly* checkbox allows you to enter *Relative Permeability* (both *Format* and *Value*) directly. Use this feature if you want to either:
>
> - enter values independent of the *Material Properties* database (the material name will default to 'Noname' when it appears in the Parts table), or
> - edit the *Relative Permeability* of a user-provided material in the *Material Properties* database.

You can also add materials to the *Material Properties* database by selecting 'Add Your Own Material' in the *Material* drop-down box. This will bring up the *Material Properties* dialog discussed in Section 3.12.3. (It can also be accessed via the *Tools->Material Properties* menu.)

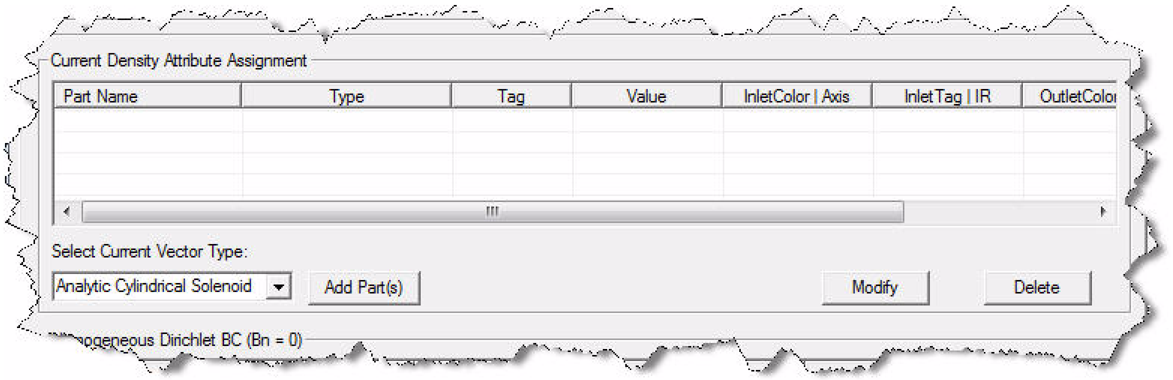
3.8.3.2 Current Density Attribute Assignment

The *Current Density Attribute Assignment* section on the *Magnetostatics* tab (Figure 43) enables the assignment of current densities to parts in the geometry model that are conductors.

Current densities can be defined as one of three types:

- a Uniform Current Density (used for straight conductors),

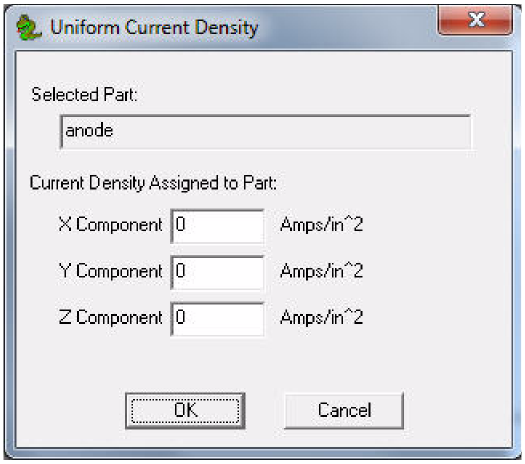
- an Analytic Cylindrical Solenoid (uniform cylindrical thickness)
- a Directional Current (used for open conductors or for solenoids with more complex shapes or nonuniform cross section)


Current Density Attribute Assignment
Part Name
Type
Tag
Value
InletColor | Axis
InletTag | IR
OutletColor
Add Part(s)
Modify
Delete
...ogeneous Dirichlet BC (Bn = 0)

Select Current Vector Type: Analytic Cylindrical Solenoid

FIGURE 43. *Current Density Attribute Assignments* table (on the *Magnetostatic* tab of the *Attribute Assignments Properties dialog*)

Uniform Current Density


Uniform Current Density
Current Density Assigned to Part:
OK
Cancel

Selected Part: anode

X Component 0 Amps/in^2

Y Component 0 Amps/in^2

Z Component 0 Amps/in^2

FIGURE 44. Uniform Current Density dialog

This option should be used for straight conductors only. By definition, a part with a uniform current density can only be a straight conductor because the current density is uniform in both magnitude and direction.

To define a part with a uniform current density, set the *Select Current Vector Type* drop down box (located beneath the *Current Density Attribute Assignment* table) to ‘Uniform Current Density’ and click the *Add Part(s)* button. The main window containing the geometric model will become active. Select the desired part by pressing Ctrl-Left mouse button. Click Enter to complete the part selection. The dialog box shown in Figure 44 will appear. Enter the values for the three components of the current density. Click *OK* to complete the definition. The part will appear in the *Current Density Attribute Assignment* table.

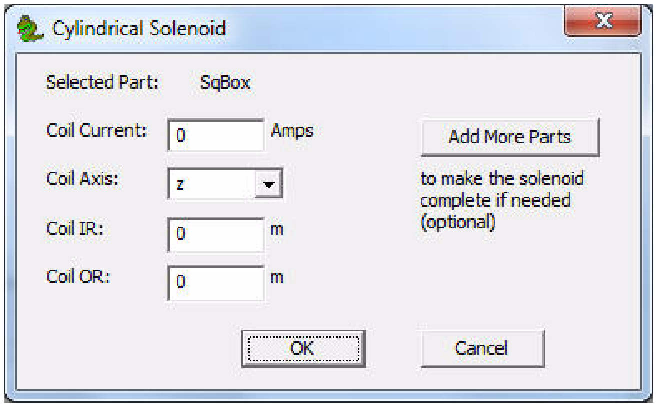
Analytic Cylindrical Solenoid

This option should be used for solenoid cylinders of uniform thickness. The geometry model for the solenoid must consist of either:

- a single cylinder of uniform thickness, or

- two parts that make up a uniformly thick cylinder and intimately in contact; each part can be any angular fraction of the cylinder cross section (e.g. two 180 degree halves, or a 90 degree piece and 270 degree piece of a cylinder, etc.)

To define a part as an analytic cylindrical solenoid, set the *Select Current Vector Type* drop down box to *Analytic Cylindrical Solenoid* and click the *Add Part(s)* button next to it. This will bring up the main window containing the geometric model. Select the desired part by pressing Ctrl-Left mouse button. Click *Enter* to complete the part selection. The dialog box shown in Figure 45 will appear. Enter the values for the *Coil Current*, select the *Coil Axis*, and set the *Coil IR* (inner radius) and the *Coil OR* (outer radius).



Selected Part: SqBox
Coil Current: 0 Amps
Coil Axis: z
Coil IR: 0 m
Coil OR: 0 m

to make the solenoid complete if needed (optional)

FIGURE 45. Cylindrical Solenoid dialog

If the chosen part represents the complete cylinder, click *OK* to complete the definition. If the cylindrical solenoid is composed of multiple pieces (as described above), use the *Add More Parts* button to select the other pieces to complete the cylinder. Click *OK* to complete the definition. The item will appear in the *Current Density Attribute Assignment* table.

Directional Current

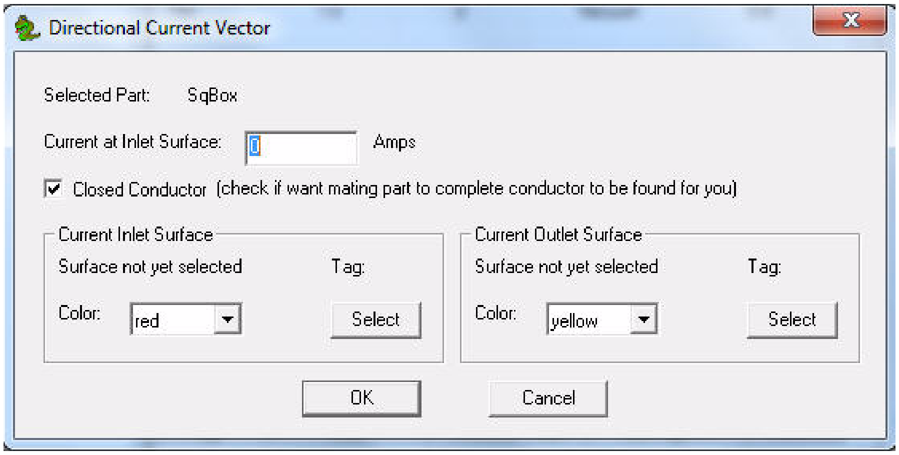
The *Directional Current* option enables you to specify a current with a uniform magnitude but directionally varying along a conductor. This option requires an inlet face (to which the incoming current is normal) and an outlet face (to which the outgoing current is also normal).

A directional current vector can be defined as open or closed. An open directional current vector should be used when the inlet and outlet faces do not overlap (e.g., a horseshoe or a C-Core conductor). A closed directional current vector should be used for conductors with varying cross section. (Solenoid cylinders of uniform thickness should use the *Analytic Cylindrical Solenoid* option discussed above rather than the Directional Current option. The Analytic Cylindrical Solenoid option is computationally faster and more accurate.)

Note that a closed conductor does not physically have an inlet or outlet, however an inlet and outlet must be specified so that boundary conditions can be prescribed for solving the current density sub-problem. For this option to work properly as a closed conductor, the geometry model must consist of two connecting parts (e.g., for a nonuniform cross section cylinder, the conductor should consist of two 180 degree halves, or a 90 degree piece and 270 degree piece of a cylinder, etc.)

To define a part as having a directional current vector, set the *Select Current Vector Type* drop down box to ‘Directional Current’ and click the *Add Part(s)* button next to it. This will bring up the main window containing the geometric model.

If you are defining an open conductor, select the desired part (i.e., the single part that represents that open conductor) by pressing Ctrl-Left mouse button. Click Enter to complete the part selection and the dialog box shown in Figure 46 will appear.


Directional Current Vector

Selected Part: SqBox

Current at Inlet Surface: 0 Amps

☑ Closed Conductor (check if want mating part to complete conductor to be found for you)

Current Inlet Surface

Surface not yet selected Tag:

Color: red

Current Outlet Surface

Surface not yet selected Tag:

Color: yellow

FIGURE 46. Directional Current Vector dialog

Enter the *Current at Inlet Surface* in *Amps* and uncheck the *Closed Conductor* checkbox. To define the inlet surface, click on the *Select* button in the *Current Inlet Surface* section. This will bring up the main window containing the geometry model. Select the desired surface by pressing Ctrl-Left mouse button. Click enter to complete the selection. The *Directonal Current Vector* dialog box will reappear. Set the color for the inlet surface using the *Color* dropdown box. Repeat the same process for the outlet surface using the *Select* button and *Color* dropdown box in the *Current Outlet Surface* section. Click *OK* to complete the directional current vector definition. The part and surfaces will appear as an entry in the *Current Density Attribute Assignment* table.

If you are defining a closed conductor (solenoid), the process is similar. However, when you initially select the desired part, you select just one of the two parts representing the solenoid. On the *Directional Current Vector* dialog, leave the *Closed Conductor* check box checked. Pick one face for the inlet and the opposite end as the outlet to specify the current direction. Because the checkbox is checked, BoaGui will automatically find the other half and complete the current assignment for you. (Note that, alternatively, you could achieve the same result by defining each half separately as having an open directional current. However, if you do this you must make sure the currents flowing in both parts are defined in the same direction.)

To modify any item in the *Current Density Attribute Assignment* table once it is defined, select it in the table and click the *Modify* button. The appropriate dialog will reappear and you can edit the values. To delete a current density definition, select it and click the *Delete* button.

3.8.3.3 Homogeneous Dirichlet Boundary Condition Assignment

The *Homogeneous Dirichlet BC (BN = 0)* section on the *Magnetostatics* tab (Figure 47) enables you to specify surfaces for which the normal component of the magnetic flux density, Bn, will be set to zero. *{TBD: ask Thuc or Mike to elaborate on when it makes sense to do this.}*

To add such a surface, click on *Add* button beneath the *Homogeneous Dirichlet BC (BN = 0)* table. This will bring up main window containing the geometric model. Find the desired surface and select it by pressing Ctrl-Left mouse button. Click enter to select. The dialog shown in Figure 48 will appear. Enter the desired *Name* (if editable) and select a *Color.* (Remember, the *Name* field will not be editable for any surface that already has a name in the CAD model.) Click *OK* to complete the assignment. The surface will appear as an entry in the table.

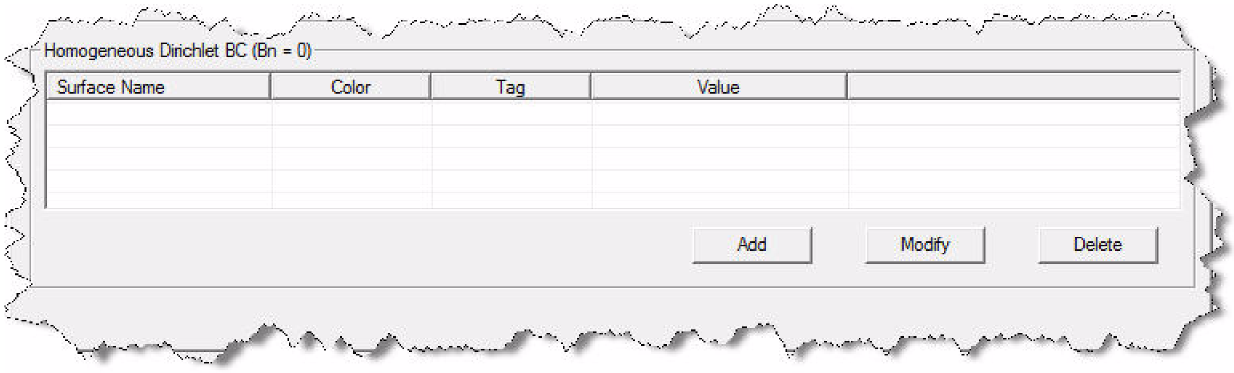
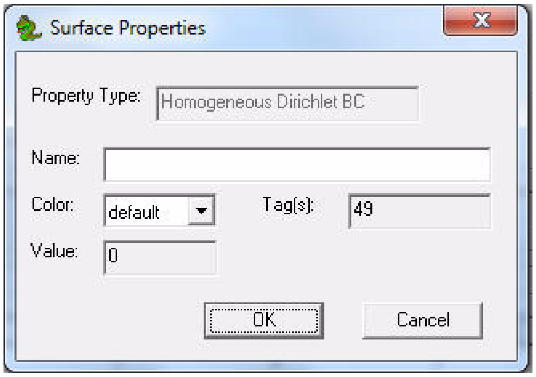


FIGURE 47. *Homogeneous Dirichlet BC (BN = 0)* table (on the *Magnetostatic* tab of the *Attribute Assignments Properties dialog*)

To modify any item in the *Homogeneous Dirichlet BC (BN = 0)* table once it is defined, select it in the table and click the *Modify* button. The above dialog will reappear and you can edit the values. To delete a surface from the table, select it and click the *Delete* button.

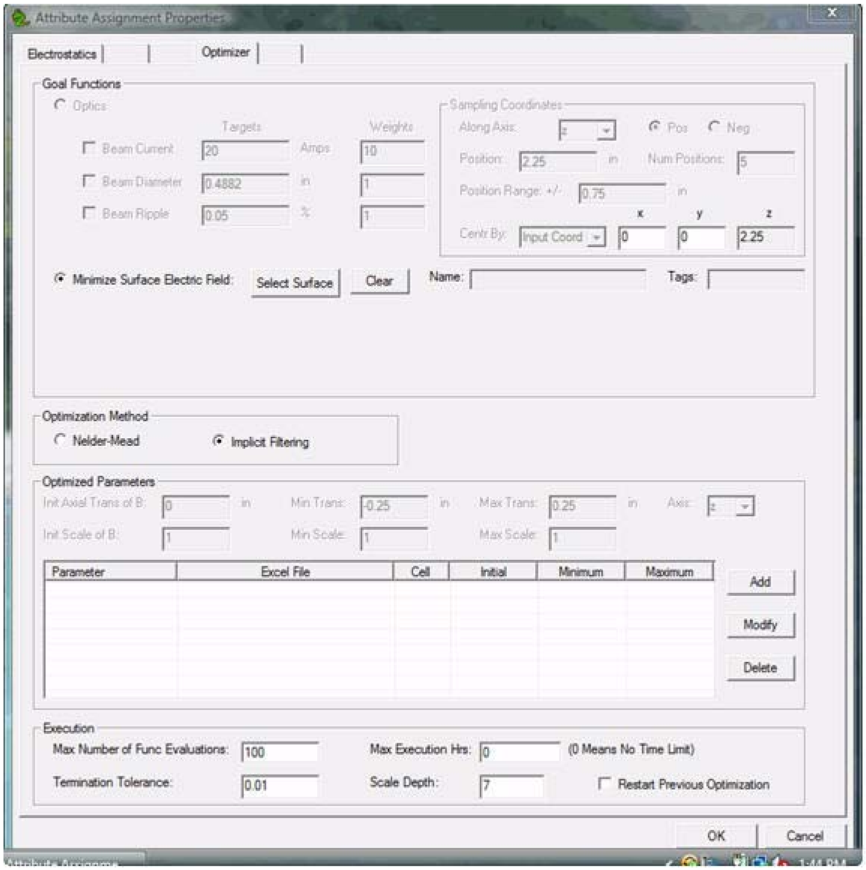


Property Type: Homogeneous Dirichlet BC
Name:
Color: default
Tag(s): 49
Value: 0

FIGURE 48. Homogeneous Dirichlet BC (BN = 0) Assignment dialog

3.8.4 Optimizer Attributes

The *Optimizer* tab, shown in Figure 49 is enabled whenever the **Analysis** sub-menu is set to **Optimizer** (rather than **Single Case**) Use it to make the attribute assignments required for optimization runs. It contains four sections. The top section (*Goal Functions*) is used to specify the goals for the optimization. The *Optimization Method* defines the method used for the optimization calculations. The *Optimized Parameters* section defines the parameters to be optimized, and the bottom section defines the *Execution* parameters for the optimization run.



Goal Functions

○ Optics

	Targets		Weights
☐ Beam Current	20	Amps	10
☐ Beam Diameter	0.4882	in	1
☐ Beam Ripple	0.05	%	1

Sampling Coordinates
Along Axis: z ● Pos ○ Neg
Position: 2.25 in Num Positions: 5
Position Range: +/- 0.75 in
Centr By: Input Coord x: 0 y: 0 z: 2.25

● Minimize Surface Electric Field: Name: Tags:

Optimization Method
○ Nelder-Mead ● Implicit Filtering

Optimized Parameters
Init Axial Trans of B: 0 in Min Trans: -0.25 in Max Trans: 0.25 in Axis: z
Init Scale of B: 1 Min Scale: 1 Max Scale: 1

Parameter	Excel File	Cell	Initial	Minimum	Maximum

Execution
Max Number of Func Evaluations: 100 Max Execution Hrs: 0 (0 Means No Time Limit)
Termination Tolerance: 0.01 Scale Depth: 7 ☐ Restart Previous Optimization

FIGURE 49. Attribute Assignment dialog box, Optimizer tab

3.8.4.1 Goal Functions

This section includes two options for the optimization's *Goal Function*: 'Optics' and 'Minimize Surface Electric Field'. For an analysis including beam optics, both options are available. For an electrostatic analysis', only the option the to minimize the surface electric field is available.

When the *Goal Function* radio button is set to 'Optics', you can specify the *Targets* for the *Beam Current*, *Beam Diameter* and/or *Beam Ripple*. (Use the checkbox in front of each to indicate which to make active.) The *Beam Ripple* can be specified only when there is a magnetic field (i.e., when **Analysis** menu is set to either 'Electro/Magnetostatic Beam' or 'Electro/Magneto Self Field Beam'). For each *Target*, specify a weight. The larger the weight, the more important that goal is in the goal function. All three targets are determined from the beam characteristics at multiple planes and then averaged for the goal function. These planes are specified in the *Sampling Coordinates* section. Choose the beam axis (the *Along Axis* dropdown box), the beam direction (the *Positive* and *Negative* radio buttons), *Position* (plane coordinate), *Number of Positions* (planes) and *Position Range*. These ranges are centered at the specified position. Thus, if the plane *Position* is 1.0" and the *Position Range* is +-0.75", then the minimum position would be 0.25" and the maximum position 1.75". The planar particle attributes are used to compute these beam characteristics, and they need a beam centroid to be transformed to cylindrical coordinates. There are three options for specifying this centroid:

- ‘All Particles’ - computes the centroid by a simple averaging of all particle (primary and secondary) coordinates on the plane,
- ‘Primary Particles’ - uses only primary particles (neglecting all secondaries),
- ‘Input Coord’ - lets the user specify the centroid coordinates manually.

Note that the first and the second option are the same if there are no secondary emissions.

When the *Goal Function* radio button is set to ‘Minimize Surface Electric Field’, use the *Select Surface* button to choose the surface on which to minimize the electric field. Only CAD named surfaces can be selected. (BoaGui will produce an error message if you do not select a named surface.) Once selected, the surface *Name* and *Tag* appear on the dialog. The *Clear* button clears the current selection. (The *Clear* button is provided as a convenience only; strictly speaking it is not actually necessary. If another surface is selected surface via the *Select Surface* button, the previous selection is automatically cleared.) Note that only one surface for field minimization can be selected at a time in the current BOA version.

3.8.4.2 Optimization Method

The current BOA version includes two iterative sampling methods for *Optimization Method*. ‘Nelder-Mead’ is the default method. It is very efficient and works well for smooth goal functions. If the goal function is noisy, and ‘Nelder-Mead’ fails to converge, use ‘Implicit Filtering’ instead.

Both ‘Nelder-Mead’ and ‘Implicit Filtering’ require many goal function evaluations. In BOA Optimizer, each function evaluation (besides computing other internal variables) runs SolidWorks once and then runs BOA until convergence.

3.8.4.3 Execution Parameters

The *Execution* section at the bottom of the dialog contains the execution parameters for the Optimizer. The parameters in this section vary slightly, depending on the selected *Optimization Method*.

- *Implicit Filtering* method - The Optimizer will terminate if the *Maximum Number of Function Evaluations* is exceeded. (This criterion is not hard set. During the course of optimization, if it is exceeded, the optimization will not stop immediately but will proceed for many more iterations before a complete stop.) In addition, the objective of the optimization is to drive the *Goal Function* values toward zero (which is unrealistic to achieve computationally) to obtain the optimal design. The *Termination Tolerance* tells the Optimizer an optimal configuration has been achieved if the goal function value is less than this tolerance. The *Maximum Execution Hrs.* tells the Optimizer to stop if the total CPU time exceeds this value. *{may not be functional yet, but should be implemented soon.}*. The *Restart Previous Optimization* check tells the Optimizer to start the optimization using the previous "optimized" parameters, which would be updated and listed in the *Optimized Parameters* sub-window. The *Scale Depth* (applicable for ‘Implicit Filtering’ only) is used to modify and scale the initial simplex when it encounters a stagnation to push the optimization forward.

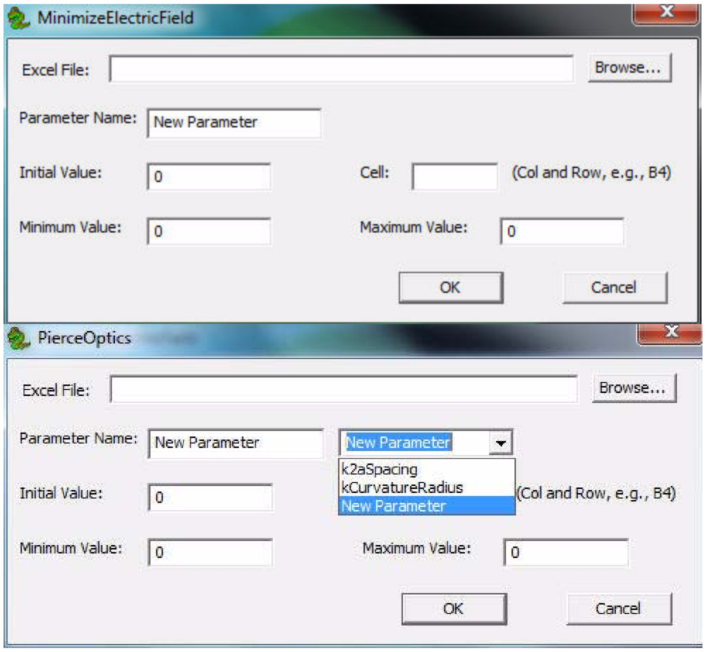
- *Nelder-Mead* method - The execution parameters are mostly the same as above. However, for Nelder-Mead, *Maximum Number of Iterations* is specified rather than *Maximum Number of Function Evaluations*. Execution terminates exactly when the number of iterations exceeds the *Maximum Number of Iterations*. Behind the scenes, the number of function evaluations in Nelder-Mead is several factors more. In addition, *Scale Depth* is not applicable.

3.8.4.4 Optimized Parameters

The *Optimized Parameters* section specifies the design parameters to be optimized.

- Magnetic Field Position and Scale - For analyses that include a magnetic filed (**Analysis** set to ‘Electro/Magnetostatic Beam’ or ‘Electro/Magneto Self Field Beam’), the top of the section specifies the initial, minimum and maximum axial translations of the field, axis of translation, and the initial, minimum and maximum field scales. (*Init Axial Trans of B*, *Min Trans*, *Max Trans*, *Axis*, *Init Scale of B*, *Min Scale* and *Max Scale*) For analyses that do not include a magnetic field (**Analysis** set to ‘Electrostatic’, ‘Electrostatic Beam’ or ‘Electrostatic Self Field Beam’), no optimization on the magnetic field position and scale is available so these fields are not enabled (as shown in Figure 49).
- Geometry Parameters - The optimized geometry parameters are specified via design tables that are originally generated by SolidWorks and then converted to Excel spreadsheets. *{Ask Mike to provide ‘how-to’ detail here.}* These design tables must be added in the *Optimized Parameters* section. Each optimized geometry parameter is represented by a coordinate in the design table. The optimization routine initially sets the value in the design table to the *Initial* value, then varied it from the specified *Minimum* to the *Maximum*. These reset design tables are then used by SolidWorks to update the geometry. In other words, these design tables are the interface between SolidWorks and the Optimizer.

  To add a geometry parameter, click the *Add* button next to the table. One of the dialogs shown in Figure 50 will appear (depending on selected *Goal Function*). Use the *Browse* button to locate the *Excel File* design table. Enter the *Parameter Name and* the *Cell* location (the column number and row number in the Excel spreadsheet, e.g., B4). (Note that when the *Goal Function* is set to ‘Optics’, the dialog includes a drop-down box with two commonly used parameters: ‘k2aspacing’ and ‘kCurvatureRadius’; BoaGui will autofill the *Parameter Name* field when one of these parameters selected). The


MinimizeElectricField

Excel File:

Parameter Name: New Parameter

Initial Value: 0

Cell: (Col and Row, e.g., B4)

Minimum Value: 0

Maximum Value: 0

PierceOptics

Excel File:

Parameter Name: New Parameter

New Parameter

Initial Value: 0

(Col and Row, e.g., B4)

Minimum Value: 0

Maximum Value: 0

FIGURE 50. Adding an Optimized Parameter (top figure is for Goal Function set to ‘Minimize Surface Electric Field’; bottom figure is for Goal function set to ‘Optics)

Minimum Value and the *Maximum Value.* Click *OK* and the geometry parameter will appear in the table in the *Optimized Parameters* section.

To modify an entry in the table, select the desired row and click the *Modify* button. To delete an entry, select the desired row and click the *Delete* button.

3.8.5 Misc Attributes

The *Temporary Thermionic Emitter Thickness Parameters (internal CCR use only)* section is for internal CCR use during the Beta testing period.

3.9 Meshing Menu

The **Meshing** pull down menu, shown in Figure 51, controls generation of the initial mesh. You must generate the initial mesh prior to executing BOA. Keep in mind that, if adaptive meshing is used, then the mesh will be adapted after the first iteration of BOA. The sections below describe each of the menu items shown in Figure 51.

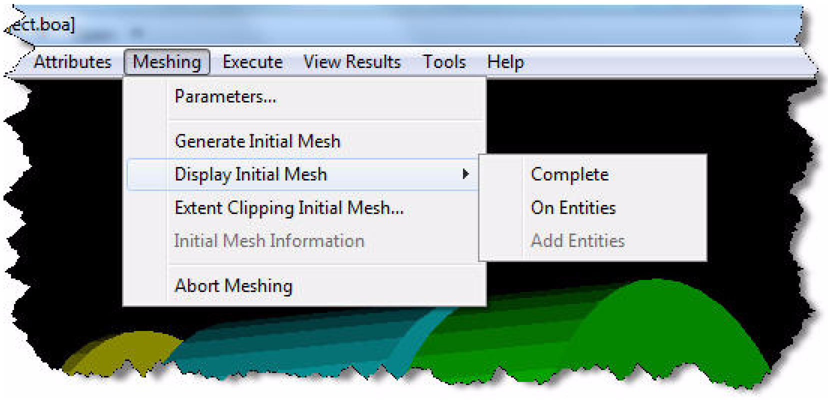
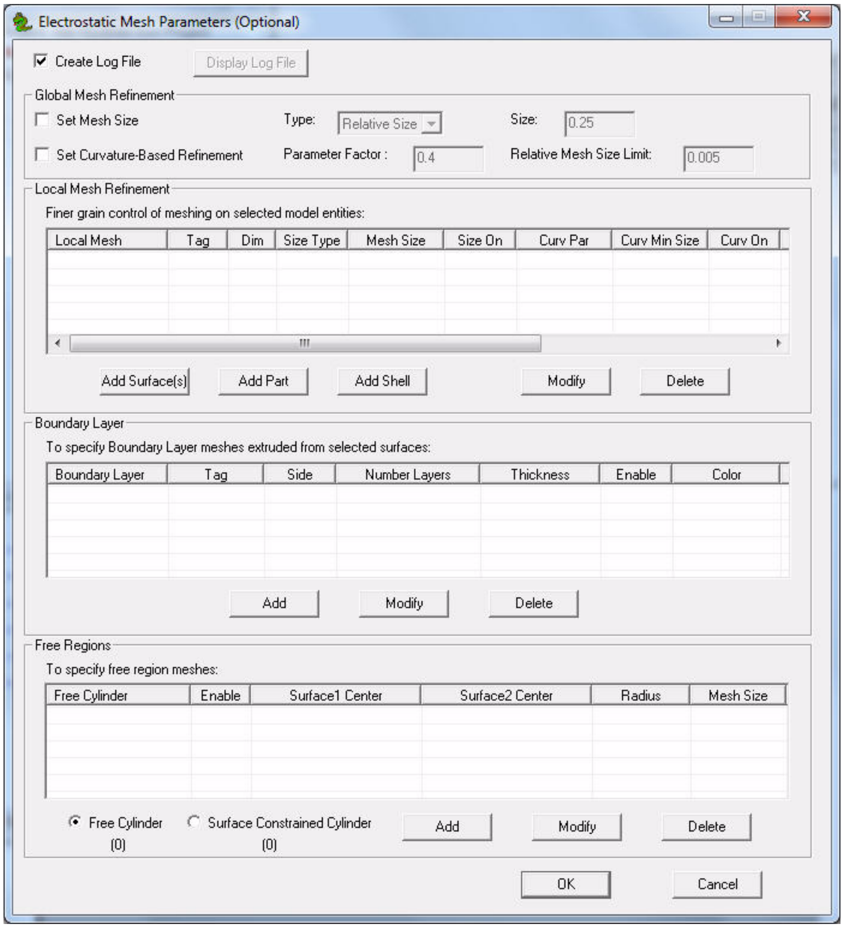

ect.boa]
Attributes
Meshing
Execute
View Results
Tools
Help
Parameters...
Generate Initial Mesh
Display Initial Mesh
Extent Clipping Initial Mesh...
Initial Mesh Information
Abort Meshing
Complete
On Entities
Add Entities

FIGURE 51. Meshing pulldown menu

3.9.1 Setting Mesh Parameters

The **Parameters** menu item opens the *Mesh Parameters* dialog box, shown in Figure 52. The primary purpose of the dialog is to provide user control over the generation of the initial mesh, as follows:


Electrostatic Mesh Parameters (Optional)
Display Log File
Global Mesh Refinement
Local Mesh Refinement
Finer grain control of meshing on selected model entities:
Local Mesh | Tag | Dim | Size Type | Mesh Size | Size On | Curv Par | Curv Min Size | Curv On
Add Surface(s)
Add Part
Add Shell
Modify
Delete
Boundary Layer
To specify Boundary Layer meshes extruded from selected surfaces:
Boundary Layer | Tag | Side | Number Layers | Thickness | Enable | Color
Add
Modify
Delete
Free Regions
To specify free region meshes:
Free Cylinder | Enable | Surface1 Center | Surface2 Center | Radius | Mesh Size
Add
Modify
Delete
OK
Cancel

☑ Create Log File

☐ Set Mesh Size Type: Relative Size Size: 0.25

☐ Set Curvature-Based Refinement Parameter Factor : 0.4 Relative Mesh Size Limit: 0.005

● Free Cylinder (0) ○ Surface Constrained Cylinder (0)

FIGURE 52. Meshing Parameters dialog box

- The *Global Mesh Refinement* section allows you to specify the global parameters used for the initial mesh generation.
- The *Local Mesh Refinement* section lets you specify mesh parameters to be used for specific entities in the geometric model.
- The *Boundary Layer* section lets you specify mesh parameters for regions that extrude from specified surface in the model.
- The *Free Region* section lets you specify mesh parameters for user-defined cylindrical regions anywhere within the bounding box of the model.

Note that if adaptivity is turned on (i.e., when *Max Number of Adaptivity Passes* is set to > 0 on the *Execute BOA* dialog), subsequent adaptive meshes can be finer, but never coarser, than the initial mesh, as specified on this dialog.

In addition to the meshing parameters, this dialog provides the option to generate a log file when the mesh routine executes. The *Create Log File* checkbox defaults to checked (i.e., create log file). The log file contains statistics associated with the meshing routine, such as number of nodes, number of meshes, number of surfaces, etc. Typically there is no need to view the log file. It is available primarily for diagnostic purposes. To view the log file, click the *Display Log File* button.

3.9.1.1 Global Mesh Refinement

The *Global Mesh Refinement* section allows you to control the global parameters used for the initial mesh generation. It includes two types of refinements: *Mesh Size* and *Curvature-based Refinement*.

> *Mesh Size* - When the *Set Mesh Size* checkbox is checked, the *Type* and the *Size* controls next to it become enabled. The *Type* dropdown box can be set to ‘Relative Size’ or ‘Absolute Size’. With ‘Absolute Size’, enter the actual physical mesh size in the units designated for the analysis. (E.g., if the geometry model has the unit of inches, enter the absolute mesh

size in inches.) ‘Relative Size’ is the mesh size relative to the largest edge of the box containing the entity to be meshed. The value must be greater than 0 and less than or equal to 1. (0 is not a valid size.)

Note that a smaller mesh size translates into a denser mesh. For structured meshes, reducing the mesh size by half increases the mesh density by roughly a factor of 8 (i.e, 2^3). This rule of thumb is less accurate for more unstructured meshes.

If the *Set Global Mesh Size* checkbox is unchecked, then the meshing routine defaults to a relative mesh size of 0.25.

Curvature-based Refinement - *Curvature-based Refinement* allows the mesh to be refined automatically to match the curvature of the entities in the geometric model. Checking the *Set Curvature-based Refinement* checkbox enables the two controls next to it. The *Parameter Factor* is the size relative to the element size. More precisely in 2D, it is the ratio of the segment height to the arc chord, which is the element size. The *Relative Mesh Size Limit* is the minimum relative mesh size due to curvature refinement on all model entities with curved surfaces.

3.9.1.2 Local Mesh Refinement

The *Local Mesh Refinement* section enables you to control the size of the mesh for specific entities in the geometry model. The table in this section is initially blank. Use the buttons beneath the table to set up local mesh refinements for specific surfaces, parts, and shells. A shell is defined as a thin part such as the grid shown in Figure 53. (The mesh definition for a shell includes additional parameters that provide for better meshing of thin parts.)

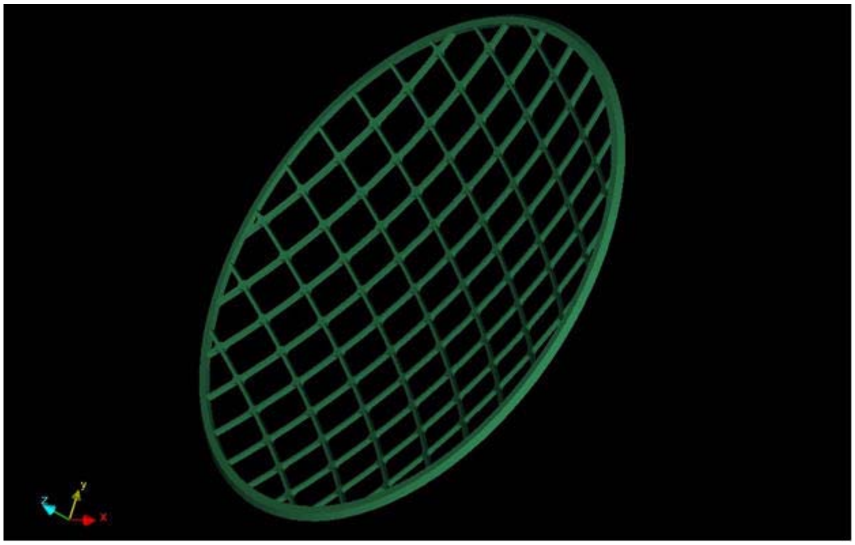
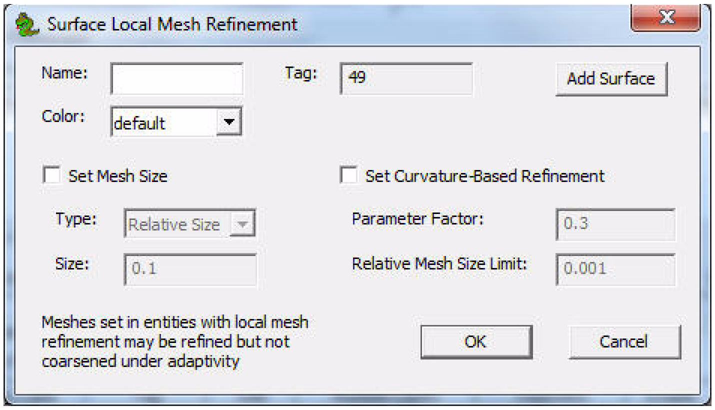


FIGURE 53. For meshing, a grid is considered a shell (a part having a small thickness)

To add a surface (or set of surfaces) to the *Local Mesh Refinements* table, click on the *Add Surface(s)* button. This will bring up the main window containing the geometric model. Find the desired surface and select it by pressing Ctrl-Left mouse button. Click enter to select. The dialog shown in Figure 54 will appear.


Surface Local Mesh Refinement

Name:

Tag: 49

Color: default

☐ Set Mesh Size

☐ Set Curvature-Based Refinement

Type: Relative Size

Parameter Factor: 0.3

Size: 0.1

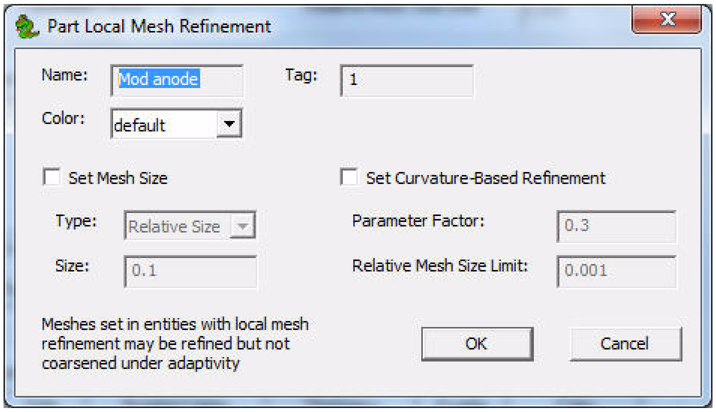
Relative Mesh Size Limit: 0.001

Meshes set in entities with local mesh refinement may be refined but not coarsened under adaptivity

FIGURE 54. Surface Local Mesh Refinement

Enter the desired *Name* (if editable) and select a *Color*. (The *Name* field will not be editable for any surface that already has a name in the CAD model.) Set the *Mesh Size* parameters (*Type* and *Size*) and the *Curvature-based Refinement* parameters (*Parameter Factor* and *Relative Mesh Size Limit*) for the surface. (These parameters are defined in the *Global Mesh Refinement* section above. Here they have a similar meaning, however they apply only to the surface(s) specified.) To add more surfaces to the entry, use the *Add Surface* button. (This option is only available for surfaces that are not named in the model.) Click *OK* to complete the assignment. The surface(s) will appear as an entry in the table.

To add a part to the *Local Mesh Refinement* table, use the *Add Part* button. The process for adding a part is nearly identical to the process for adding a surface, as described above. The dialog, shown in Figure 55, includes the same parameters discussed above.


Part Local Mesh Refinement

Name: Mod anode

Tag: 1

Color: default

☐ Set Mesh Size

☐ Set Curvature-Based Refinement

Type: Relative Size

Parameter Factor: 0.3

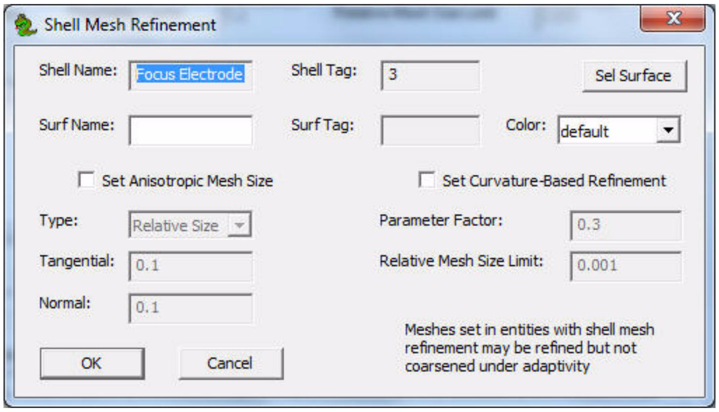
Size: 0.1

Relative Mesh Size Limit: 0.001

Meshes set in entities with local mesh refinement may be refined but not coarsened under adaptivity

FIGURE 55. Part Local Mesh Refinement

To add a shell to the *Local Mesh Refinements* table, click on the *Add Shell* button. This will bring up the main window containing the geometric model. Move the cursor to the part you want defined as a shell and select it by pressing Ctrl-Left mouse button. Click *Enter* to complete the selection. The dialog shown in Figure 56 will appear. Enter the desired *Shell Name* (if editable). (The *Shell Name* field will not be editable for any part that already has a name in the CAD model.)



Shell Name: Focus Electrode
Shell Tag: 3
Surf Name:
Surf Tag:
Color: default

☐ Set Anisotropic Mesh Size
☐ Set Curvature-Based Refinement

Type: Relative Size
Parameter Factor: 0.3
Tangential: 0.1
Relative Mesh Size Limit: 0.001
Normal: 0.1

Meshes set in entities with shell mesh refinement may be refined but not coarsened under adaptivity

FIGURE 56. Shell Mesh Refinement

With shell mesh refinement, you can specify the mesh size across the shell thickness (*Normal*) as well as along its surfaces (*Tangential*). These parameters become active when you check the *Set Anisotropic Mesh Size* checkbox. In order for the GUI to determine the unit vector normal to the shell surfaces, you must specify one of the shell surfaces. Click on the *Sel Surface* button to bring up the main window containing the geometric model. Move the cursor to the desired surface (on the shell), press Ctrl-Left mouse button and *Enter* to select. The *Shell Mesh Refinement* dialog will reappear. Enter the desired *Surface Name* (if editable) and select a *Color*. (The *Surface Name* field will not be editable for any surface that already has a name in the CAD model.) You can also set the *Curvature-based refinement* parameters (same as discussed above). To complete the assignment, click *OK*. The shell will appear as an entry in the table.

To modify a local mesh refinement entry once it has been defined, select it in the table and click the *Modify* button. The appropriate dialog will reappear and you can edit the values. To delete an entry from the table, select it and click the *Delete* button.

3.9.1.3 Boundary Layers

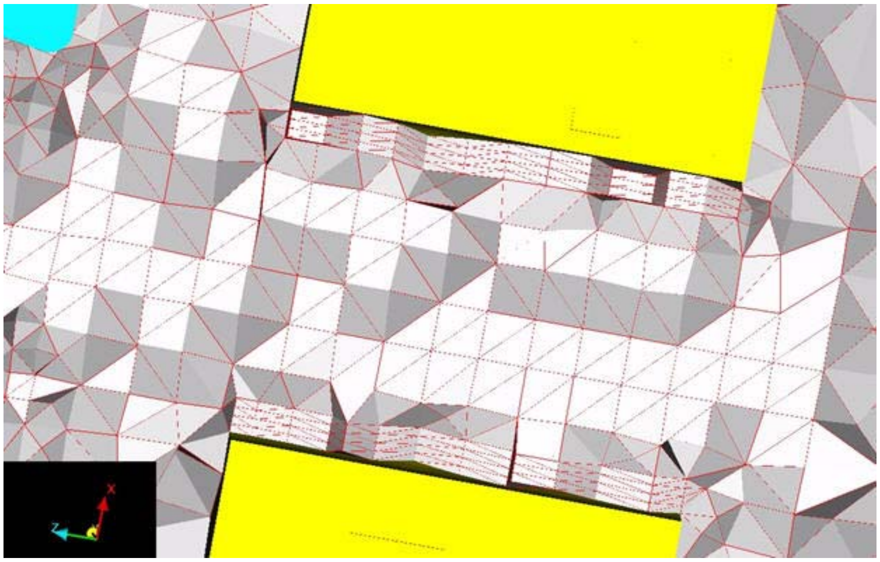
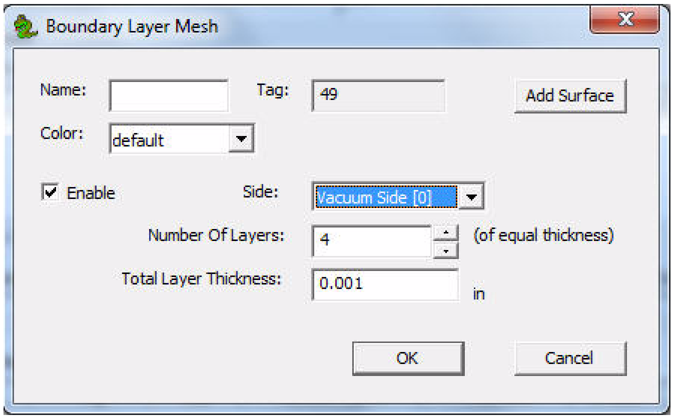


FIGURE 57. Boundary Layer Meshes extruding from an anode. Here, there are four layers of equal thickness meshes.

Boundary Layers are layers of semi-structured meshes extruding from a prescribed surface (see Figure 57). They are useful if high field fidelity is desired in the neighbor of a particular surface.

To add a *Boundary Layer Mesh*, click on the *Add* button beneath the table. This will bring up main window containing the geometric model. Find the surface from which the boundary layer should extrude and select it by pressing Ctrl-Left mouse button. Click *Enter* to select. The dialog shown in Figure 58 will appear.



Name:
Tag: 49
Color: default
☑ Enable
Side: Vacuum Side [0]
Number Of Layers: 4 (of equal thickness)
Total Layer Thickness: 0.001 in

FIGURE 58. Boundary Layer Mesh

Enter the desired *Name* (if editable) and select a *Color.* (Remember, the *Name* field will not be editable for any surface that already has a name in the CAD model.) Make sure the *Enable* check box is checked. (This checkbox allows you to disable boundary layer meshes for a run without having to delete them from the case.) Select the desired *Side* (‘Vacuum side, ‘Solid side’ or ‘Both sides’). Set the *Number of Layers* and the *Total Layer Thickness* in the analysis units. Each layer will be of equal thickness. To add more surfaces to the entry, use the *Add Surface* button. (This option is available only for surfaces that are not named in the model.) Click *OK* to complete the assignment. The surface(s) will appear as an entry in the table.

To modify a boundary layer definition once it is entered, select it in the table and click the *Modify* button. The appropriate dialog will reappear and you can edit the values. To delete an entry from the table, select it and click the *Delete* button.

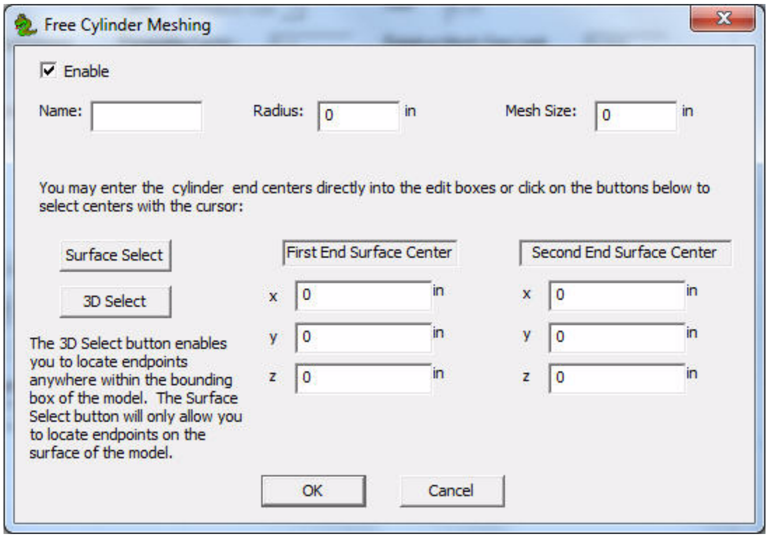
3.9.1.4 Free Regions

Free Regions are user-defined cylindrical regions for which the user can define a mesh size. One reason for adding a free region is to get better resolution around the beam area in a beam analysis. This cylindrical region can be defined in one of two ways:

- A *Free Cylinder* is defined by specifying two end face center points and a radius. The cylinder axis is the line connecting the two centers.
- A *Surface-constrained Cylinder* is defined by first selecting a surface and then defining a radius and a length. The surface center is internally computed; the cylinder axis originates from this center and is perpendicular to the constrained surface.

To add a *Free Cylinder* mesh, select the *Free Cylinder* radio button and click on the *Add* button beneath the table. This will bring up the dialog shown in Figure 59.

Enter the desired *Name* and select a *Color*. Make sure the *Enable* check box is checked. (This checkbox allows you to disable free regions for a run without having to delete them from the case.) Set the desired *Mesh Size*. Define the cylindrical region by specifying *Radius*, and the two endpoints *(First End Surface Center* and *Second End Surface Center)*. These endpoints can be entered in one of three ways:



☑ Enable

Name:

Radius: 0 in

Mesh Size: 0 in

You may enter the cylinder end centers directly into the edit boxes or click on the buttons below to select centers with the cursor:

The 3D Select button enables you to locate endpoints anywhere within the bounding box of the model. The Surface Select button will only allow you to locate endpoints on the surface of the model.

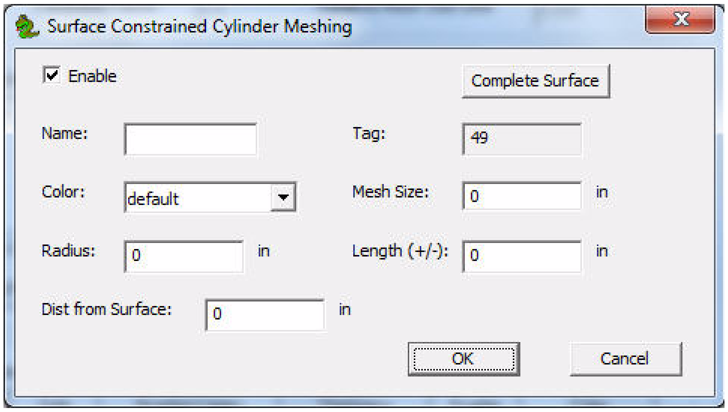
	First End Surface Center		Second End Surface Center	
x	0	in	0	in
y	0	in	0	in
z	0	in	0	in

FIGURE 59. Free Cylinder Meshing dialog

- Direct entry allows you to enter the *x*, *y*, and *z* components of each point directly in the edit fields provided.
- Surface selection allows you to select two points anywhere on the surface of the model. Click the *Surface Select* button and the main window containing the geometry model will become active. Use the Ctrl-Left mouse button to select the first and second end points. Only points on surfaces can be selected. A small dot will appear for each endpoint and the xyz coordinates will automatically appear in the fields provided on the *Free Cylinder Meshing* dialog.
- 3D select allows you to select two endpoints anywhere within the bounding box of the model. Click the *3D Select* button and the main window containing the geometry model will become active. A line widget with two endpoints will appear. Use the Ctrl-Left mouse button to move the endpoints to their desired locations. To complete the selection, click *3D Select* on the *Free Cylinder Meshing* dialog. The xyz coordinates will automatically appear in the fields provided.

Click *OK* to complete the assignment. The free region will appear as an entry in the table.

To add a *Surface-constrained Cylinder* mesh, select the *Surface-constrained Cylinder* radio button and click the *Add* button. This will bring up the main window containing the geometry model. Find the surface on which to constrain the desired cylindrical region and select it by pressing Ctrl-Left mouse button. Click *Enter* to complete the selection. The dialog shown in Figure 60 will appear.


Surface Constrained Cylinder Meshing
Complete Surface
OK
Cancel

☑ Enable

Name:

Tag: 49

Color: default

Mesh Size: 0 in

Radius: 0 in

Length (+/-): 0 in

Dist from Surface: 0 in

FIGURE 60. Free Cylinder Meshing

Enter the desired *Name* (if editable) and select a *Color*. (Remember, the *Name* field will not be editable for any surface that already has a name in the CAD model.) Make sure the *Enable* check box is checked. (This checkbox allows you to disable free regions for a run without having to delete them from the case.) Set the desired *Mesh Size*. Define the cylindrical region by specifying its *Radius* and *Length (+/-)*. Specifying a positive length means the cylinder will be generated away from the solid part to which the surface is adjacent. Specifying a negative length means the cylinder will be generated into the part. You can also specify a nonzero *Distance from Surface* and the meshes will be created the specified distance away from the constrained surface.

The *Complete Surface* button is provided for cases where the surface is broken into multiple pieces and does not have a CAD name. (A CAD named surface, even if split into multiple segments, could be selected as one entity, obviating the need for using this button.) For such a case, use the *Complete Surface* button to add the other pieces to complete the selection. Click *OK* to complete the assignment. The surface-constrained cylinder will appear as an entry in the table.

To modify a free region once it is defined, select it in the table and click the *Modify* button. The appropriate dialog will reappear and you can edit the values. To delete an entry from the table, select it and click the *Delete* button.

3.9.2 Generating and Displaying the Initial Mesh

The next set of items on the **Meshing** pulldown menu (see Figure 61) control the generation and display of the initial mesh.

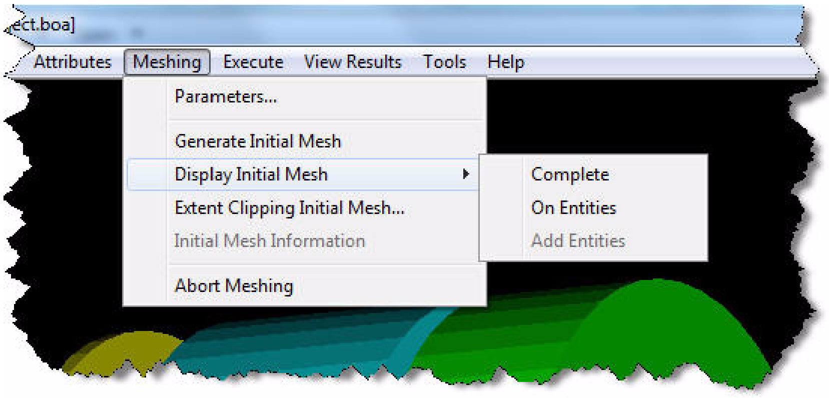
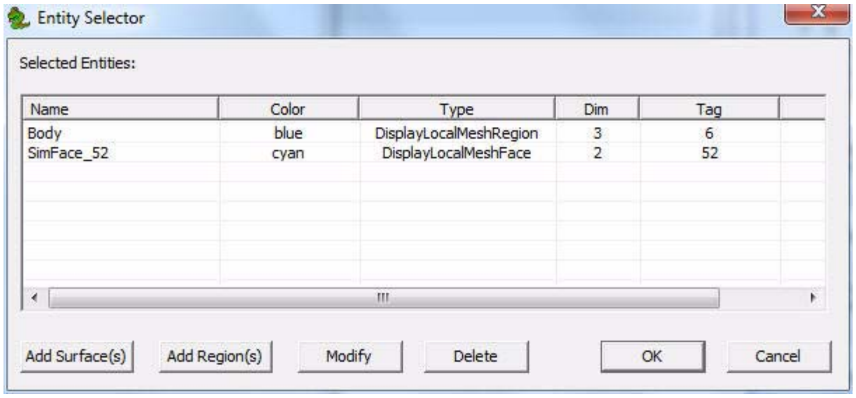

ect.boa]
Attributes
Meshing
Execute
View Results
Tools
Help
Parameters...
Generate Initial Mesh
Display Initial Mesh
Extent Clipping Initial Mesh...
Initial Mesh Information
Abort Meshing
Complete
On Entities
Add Entities

FIGURE 61. Meshing pulldown menu

3.9.2.1 Generation of Initial Mesh

The **Generate Initial Mesh** menu item initiates generation of the initial mesh. A DOS window will appear indicating progress during the mesh generation process. Once the process is completed, hit any key to close the DOS window and return control back to the user interface. You must generate the initial mesh before you execute BOA.

3.9.2.2 Display of Initial Mesh



Entity Selector

Selected Entities:

Name	Color	Type	Dim	Tag
Body	blue	DisplayLocalMeshRegion	3	6
SimFace_52	cyan	DisplayLocalMeshFace	2	52

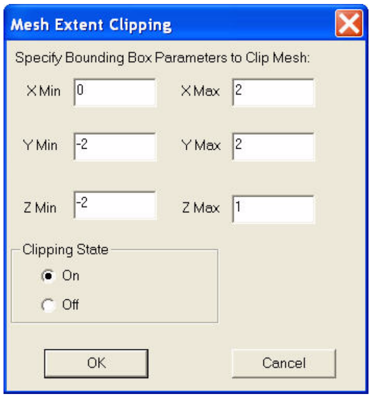
FIGURE 62. Entity Selector Dialog

The **Display Initial Mesh** menu item turns the display of the initial mesh on and off in the main window. As shown in Figure 61, this item contain three sub-menus. The **Complete** item toggles the complete mesh on and off. The **On Entities** item toggles the mesh on and on for user-selected entities. The **Add/ Remove Entities** menu item brings up the *Entity Selector* dialog shown in Figure 62. Use the *Add Surface(s)* and *Add Region(s)* buttons to add the desired surfaces and regions. *{TBD: get definition of Region. in some cases it allows you to select whole parts and other times just a subsection of Part. Get explanation.}*

When the mesh is displayed, the menu item (either *Complete* or *On Entity*) will appear with a check mark in front of it. Note that you will only be able to see the mesh on the outer boundaries of the problem domain (i.e. on the outer boundaries of the background part). To see the internal mesh structure, use *Extent Clipping* described in the next section.

3.9.2.3 Extent Clipping for Initial Mesh

The **Extent Clipping Initial Mesh** menu item brings up the *Mesh Extent Clipping* dialog box shown in Figure 63. This feature allows you to remove sections of the mesh in order to view the internal structure. When the *Clipping State* radio buttons is set to Off, the graphics window displays the mesh on the outer boundaries of the problem domain. When the radio button is set to On, the XYZ ranges in the *Specify Bounding Box Parameters to Clip Mesh* section are used to determine which area of the mesh to display. Keep in mind that you can choose to include display of all or selected parts in the model geometry using the *Visible* checkbox on the *Electrostatics* tab of the *Attribute Assignment* window.



Mesh Extent Clipping

Specify Bounding Box Parameters to Clip Mesh:

X Min 0
X Max 2
Y Min -2
Y Max 2
Z Min -2
Z Max 1

Clipping State
● On
○ Off

FIGURE 63. Mesh Extent Clipping dialog

A sample display is shown in Figure 64. Here an initial mesh was generated for a Pierce electron gun. Parts of the gun were turned off in the *Attribute Assignment* window to allow visualization of the interior. The user specified that mesh elements with X values less than 0 should be removed. Since whole elements are removed, the clipped surface can appear rather strange, as shown in the figure. Never the less, information on the mesh size and location can be determined.

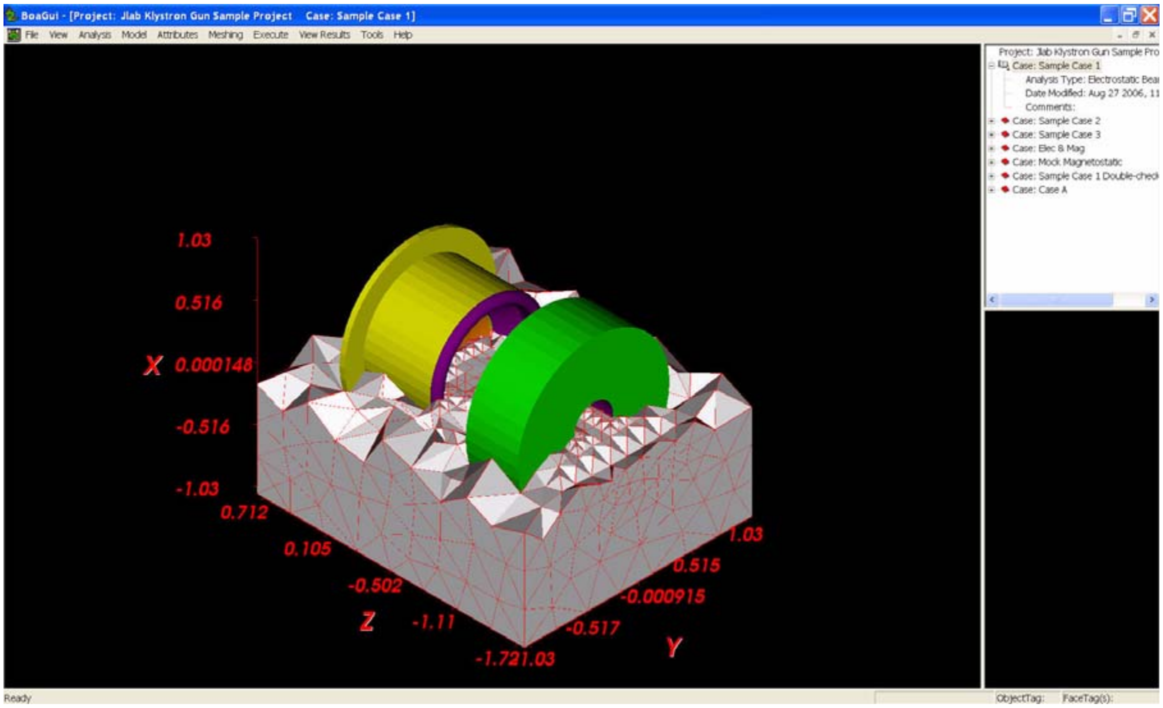
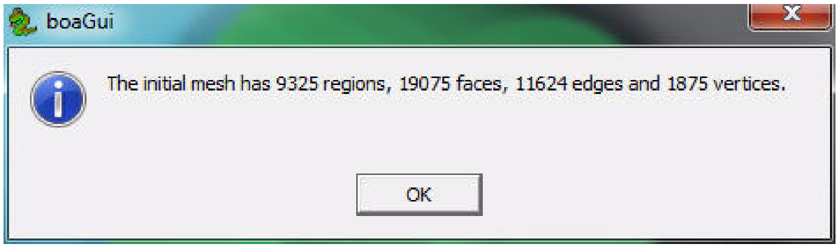

BoaGui - [Project: Jlab Klystron Gun Sample Project Case: Sample Case 1]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
Project: Jlab Klystron Gun Sample Pro
Case: Sample Case 1
Analysis Type: Electrostatic Bea
Date Modified: Aug 27 2006, 11
Comments:
Case: Sample Case 2
Case: Sample Case 3
Case: Elec & Mag
Case: Mock Magnetostatic
Case: Sample Case 1 Double-check
Case: Case A
1.03
0.516
X 0.000148
-0.516
-1.03
0.712
0.105
-0.502
Z -1.11
-1.721.03
1.03
0.515
-0.000915
-0.517
Y
Ready
ObjectTag:
FaceTag(s):

FIGURE 64. Extent clipping parallel to Y-Z plane at X=0.

3.9.2.4 Initial Mesh Information

This **Initial Mesh Information** menu item brings up the information dialog shown in Figure 65. It provides statistic for the initial mesh, including:

- number of regions,
- number of faces,
- number of edges, and
- number or vertices.



The initial mesh has 9325 regions, 19075 faces, 11624 edges and 1875 vertices.

FIGURE 65. Initial Mesh Information

3.9.2.5 Aborting Mesh Generation

To abort meshing while mesh generation is in progress, select the **Abort Meshing** menu item (the last item on the menu shown in Figure 61*).*

3.10 Execute Menu

The **Execute** pull down menu, shown in Figure 66, controls the execution of the BOA program. It includes options for executing a **Single Case** run, executing an **Optimizer** run, and monitoring a run (**Job Monitor Parameters** and **Monitor Job**). The sections below describe each of the menu items shown in Figure 66.

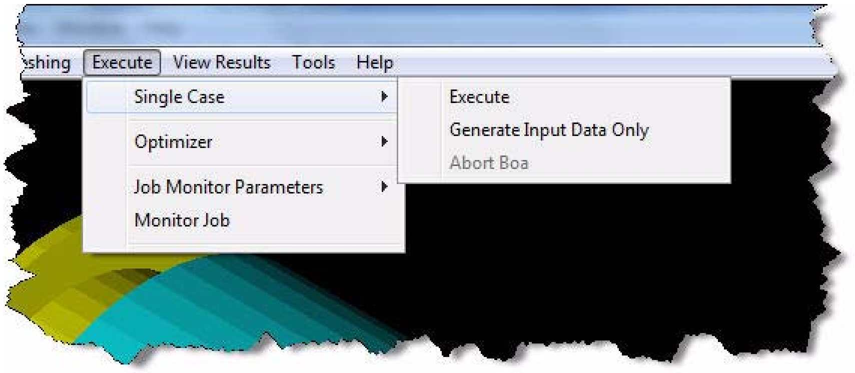


FIGURE 66. Execute pulldown menu

3.10.1 Executing a Single Case

The **Single Case** menu item opens the sub-menu shown in Figure 66. It includes:

- **Execute -** Initiates the single case execution.
- **Generate Input Data Only** - Generates only the input data set for the single case execution BOA can be executed using the command line, provided that the required input parameters and files are provided. This allows the user to make rapid or systematic changes to the input without invoking the graphical user interface. For advanced users, this can result in a significant increase in efficiency. The necessary command line arguments for executing BOA are provided in the Appendix. The naming convention for the input data file is CaseName.att (where CaseName is the name of the current case). The file will be placed in the Case directory.
- **Abort Boa** - This item allows you to abort a simulation in progress. This menu item is intended primarily for Beam Optics simulations. (Electrostatic and magnetostatic simulations run quickly so there is little need for an Abort function.) Prior to aborting, an informational message will report the last iteration available for post-processing and give you the option to cancel or continue with the termination. The last iteration available for post-processing depends on the settings in the *Beam Optics Parameters* section of the *Execute Boa* dialog.

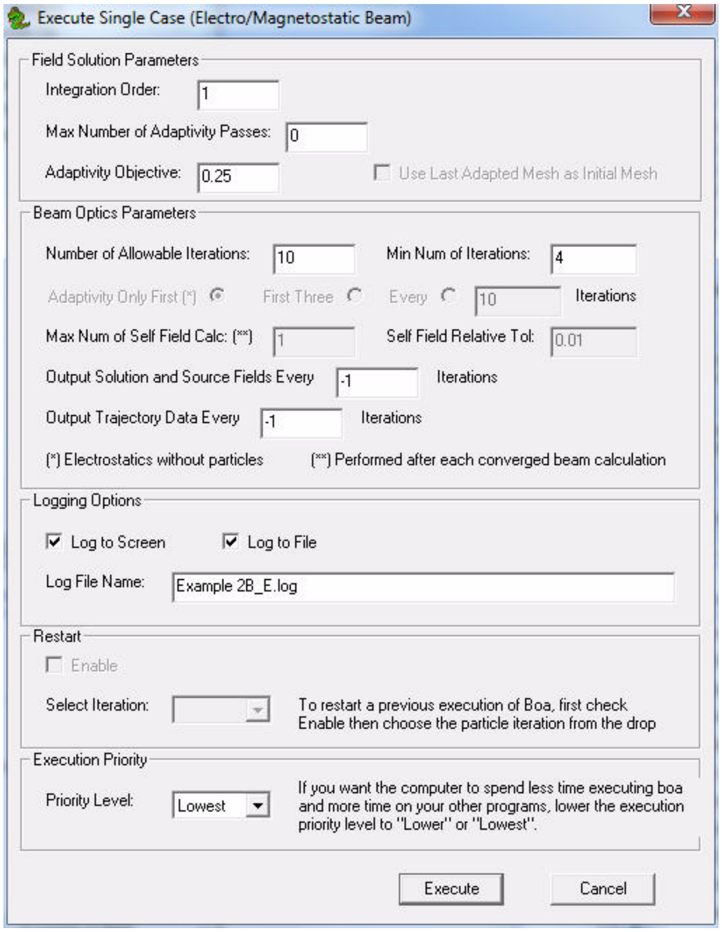
The first two menu items initiate the dialog shown in Figure 67. It includes sections for *Field Solution Parameters*, *Beam Optics Parameters Logging Options*, *Restart* and *Execution Priority*. These sections are discussed below.

When you click *Execute* a DOS window will appear indicating progress during BOA execution. Once the process is completed, hit any key to close the DOS window and return control back to the user interface.

3.10.1.1 Field Solution Parameters

The *Field Solution Parameters* section of the *Execute BOA* dialog includes:

Integration Order - This parameter sets the order of the finite element. Enter either 1 (linear) or 2 (quadratic). Quadratic integration is more accurate; however, it can take 3 to 4 times longer to reach a solution. The default value is 1.



Field Solution Parameters

Integration Order: 1

Max Number of Adaptivity Passes: 0

Adaptivity Objective: 0.25

☐ Use Last Adapted Mesh as Initial Mesh

Beam Optics Parameters

Number of Allowable Iterations: 10

Min Num of Iterations: 4

Adaptivity Only First (*) ● First Three ○ Every ○ 10 Iterations

Max Num of Self Field Calc: (**) 1

Self Field Relative Tol: 0.01

Output Solution and Source Fields Every -1 Iterations

Output Trajectory Data Every -1 Iterations

(*) Electrostatics without particles (**) Performed after each converged beam calculation

Logging Options

☑ Log to Screen ☑ Log to File

Log File Name: Example 2B_E.log

Restart

☐ Enable

Select Iteration:

To restart a previous execution of Boa, first check Enable then choose the particle iteration from the drop

Execution Priority

Priority Level: Lowest

If you want the computer to spend less time executing boa and more time on your other programs, lower the execution priority level to "Lower" or "Lowest".

FIGURE 67. Execute dialog

Max Number of Adaptivity Passes -
This parameter sets the maximum number of times BOA will call the adaptive meshing routines for mesh refinement. Typically, this value will be 2 or 3; however, in some cases it may be advantageous to set this to a lower value to reduce the turnaround time at the cost of accuracy. This may be particularly applicable during the early stages of the design. Setting the value to 0 will result in execution of BOA using only the initial mesh. The default value is 0.

Adaptivity Objective - This parameter sets the accuracy objective for the adaptive meshing routines. Provided the number of adaptivity passes does not exceed the *Maximum Number of Adaptivity Passes*, the program will continue to invoke the mesh generation routines until the relative *Energy Error Norm Estimate* between the current iteration and the previous iteration is less than the *Adaptivity Objective*. (The *Energy Error Norm Estimate* is a global value describing the accuracy of the solution. BOA computes this value internally based on the numerical solution. It is not an input parameter. See the Appendix for details.) The default value is 0.25.

The *Use Last Adapted Mesh as Initial Mesh* checkbox allows you to start a simulation using the last adapted mesh. This allows you to reduce the analysis time if you have already done an adaptive simulation with a good mesh, and want to repeat the analysis with some minor changes such as small increase/decrease in beam voltage, different convergence criteria, etc. In such a scenario, you can just turn off adaptivity (set *Max Number of Adaptivity Passes* = 0) and start with an already "good" initial mesh.

3.10.1.2 Beam Optics Parameters

The *Beam Optics Parameters* section of the *Execute BOA* dialog is active when the **Analysis** type includes beam optics. This section contains the following parameters:

Number of Allowable Iterations - This parameter is the maximum number of iterations allowed in the BOA algorithm. If the solution does not converge after the specified number of iterations, BOA will stop the analysis and exit. The default value is 10.

Min Num of Iterations - The minimum number of iterations performed in the BOA algorithm.

Adaptivity Only... Radio Buttons - These radio buttons control how often BOA calls the adaptivity routine. It is enabled when *Max Number of Adaptivity Passes* > 0. Options include performing adaptivity for:

- the first iteration only,
- the first three iterations, or
- after a user-specified number of iterations.

(Note that the first iteration does not include particles. The self-consistent method, which BOA uses to push particles, iterates on the electric field and emission current until their normalized relative errors are less than *Electric Field Convergence Criterion* and *Emission Current Convergence Criterion*, respectively, (both criteria are specified on the *Beam Optics* tab of the *Attribute Assignment Properties* dialog), or until the number of iterations exceeds the *Number of Allowable Iterations*. In the first iteration there are no particles yet, thus the electrostatic field is solved without particles before any can be pushed. Adaptive meshing is performed using the electric field. Therefore, if the first radio button is selected (adaptivity for first iteration only), no particles are involved.

Max Number of Self Field Calc and *Self Field Relative Tol*. - These parameters are enabled when the **Analysis** Type is either ‘Electrostatic/Self Field Beam’ or ‘Electro/Magneto/Self Field Beam’. With the self field calculation, the normalized relative energy norm of the self field of the previous self field iteration to the present is compared with the *Self Field Relative Tolerance*. If the normalized relative energy norm is less, the self field solution converges. Also, if the number of self field iterations exceeds the *Max Number of Self Field Calc.*, the self field solution also stops.

In summary the self field calculation algorithm is:

a. The beam without the self field is run until it converges
b. The converged beam current in (a) is used to compute the self field
c. With the just-computed self field kept frozen, the beam is run again until convergence
d. Steps (a)-(c) are repeated until the number of self field iterations exceeds the *Max Number of Self Field Calc* or the relative energy norm of the present to the previous iterated self field is less than the *Self Field Relative Tolerance*.

Output Solution and Source Fields Every {N} Iterations - This parameter controls how often the solution and source fields are output to results files. A value of -1 (the default value) means the solution and source fields are written to files for the first two iterations (the 0th[1] and 1st iterations) and for the last iteration. This is the case regardless of whether or not the solution converges.

Example: If you set this value to 3 and the simulation stops at the 14th iteration, then the solution is written to files at 0th, 1st, 4th, 7th, 10th, 13th and lastly 14th iterations. If you abort the job at the 9th iteration, you can still obtain the results of the last written iteration, which in this case would be the 7th iteration. Alternatively, if you set this value to 5 and abort at the 9th iteration, you have only the 6th iteration available. If you use the default value of -1 and abort at the 9th iteration, you have only the 1st iteration to postprocess.

Output Trajectory Data Every {N} Iterations - This parameter controls how often the trajectory data is output to results files. A value of -1 (the default value) means the trajectory data is written to a file for the first two iterations (the 0th and 1st iterations) and for the last iteration. This is the case regardless of whether or not the solution converges.

3.10.1.3 Logging Options

The *Logging Options* section provides options for showing the run log real time (*Log to Screen* checkbox) and creating a log file (*Log to File* checkbox). The log filename defaults to [Case Name]_.log. Use the *Log File Name* edit box to change the filename if desired. The log file is placed in the case directory. The log file can be accessed within BoaGui from the **View Results** pulldown menu.

3.10.1.4 Restart

The *Restart* section will be active if iterations from a previous BOA run exist. To restart a previous execution of Boa, check the *Enable* box then choose the particle iteration from the *Select Iteration* drop down box. The iterations available for selection are based on you selections for both *Output Solution and Source Fields Every {N} Iterations* and *Output Trajectory Data Every {N} Iterations*, described in the previous section.

3.10.1.5 Execution Priority

The *Execution Priority* section gives you some control over how much of your computer’s resources are used when executing BOA. If you want the computer to spend less time executing BOA and more time on your other programs, change the *Priority Level* from ‘Normal’ to ‘Lower’ or ‘Lowest’.

3.10.2 Executing an Optimizer Case

The **Optimizer** menu item... *{TBD. waiting for Optimizer example from M in order to see dialogs. (can’t run without Solidworks)}*

---

[1] The 0th iteration of a beam simulation is the very first iteration with NULL space charge. From the 1st iteration on, space charge will be present due to particle motion.

3.10.3 Monitoring a Beam Optics Run

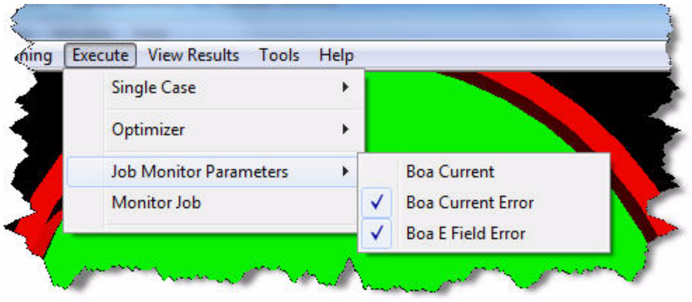
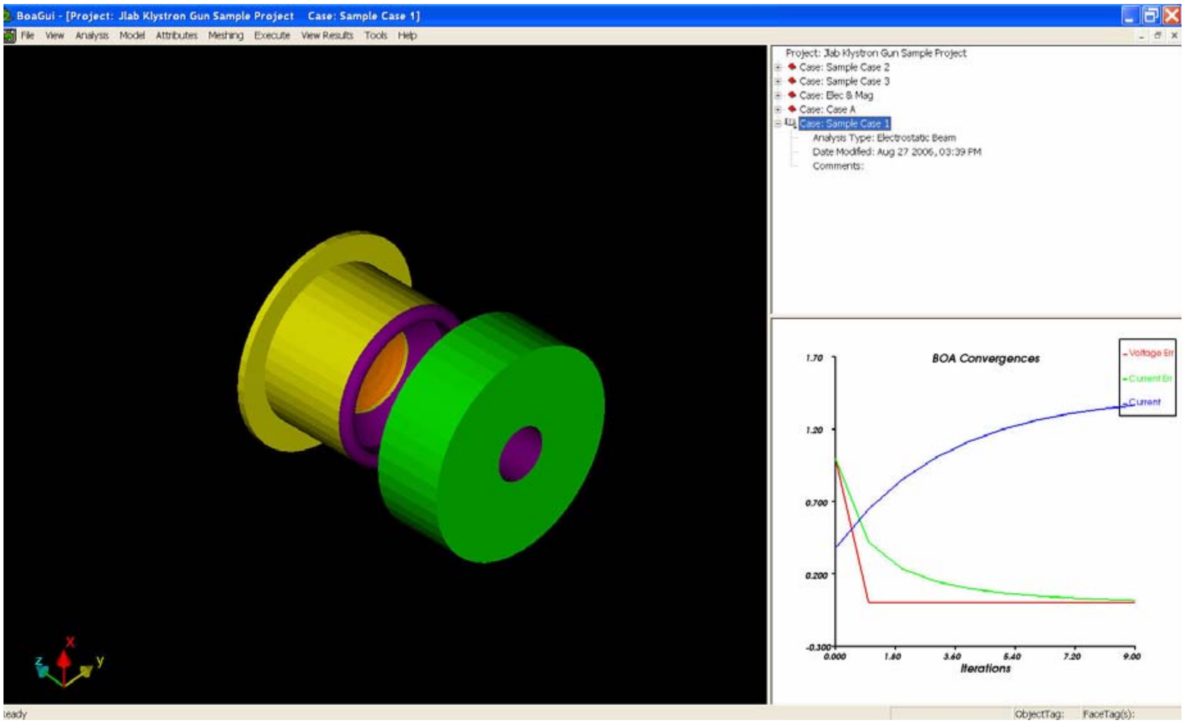

hing
Execute
View Results
Tools
Help
Single Case
Optimizer
Job Monitor Parameters
Monitor Job
Boa Current
✓ Boa Current Error
✓ Boa E Field Error

FIGURE 68. Job Monitoring Parameters

The two Job Monitoring menu items enable allow you to dynamically monitor the convergence of a beam optics simulation. As shown in Figure 68, the **Job Monitor Parameters** menu item allows you to select the desired parameters for plotting: Current, Current Error and/or Potential Error. The **Monitor Job** menu item lets you toggle the monitoring on and off. The convergence plot appears in the lower right hand region of the BoaGui as shown in Figure 69. The convergence plot is automatically updated behind the scene as the beam simulation progresses.


BoaGui - [Project: Jlab Klystron Gun Sample Project Case: Sample Case 1]

Project: Jlab Klystron Gun Sample Project
- Case: Sample Case 2
- Case: Sample Case 3
- Case: Elec & Mag
- Case: Case A
- Case: Sample Case 1
  - Analysis Type: Electrostatic Beam
  - Date Modified: Aug 27 2006, 03:39 PM
  - Comments:

File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
x
y
z
BOA Convergences
Voltage Err
Current Err
Current
1.70
1.20
0.700
0.200
-0.300
0.000
1.80
3.60
5.40
7.20
9.00
Iterations
eady
ObjectTag:
FaceTag(s):

FIGURE 69. BOAGui shown with Job Monitoring On for a beam optics simulation

3.11 View Results Menu

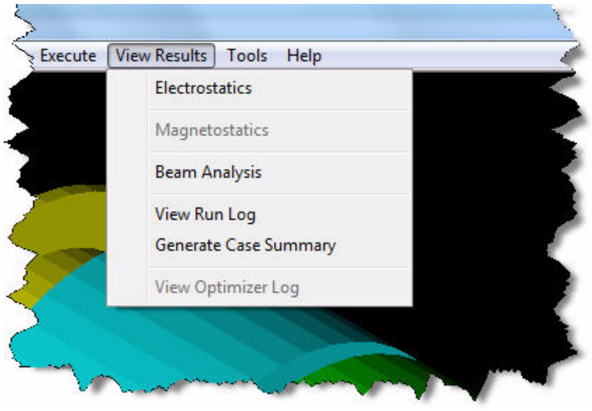
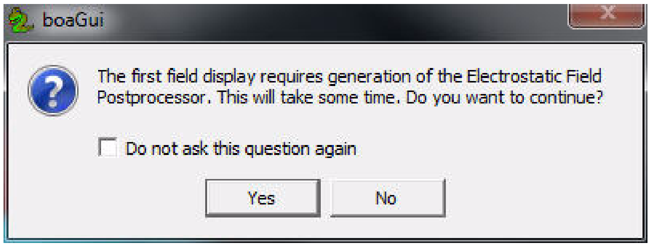

Execute
View Results
Tools
Help
Electrostatics
Magnetostatics
Beam Analysis
View Run Log
Generate Case Summary
View Optimizer Log

FIGURE 70. View Results Pull-down Menu

The **View Results** pulldown menu, shown in Figure 70, provides access to all results, including: **Electrostatics** (available for electrostatic analyses only), **Magnetostatics** (available for magnetostatic analyses only), and **Beam Analysis** (available for analyses that include beam optics). The menu also enables you to view the execution log (**View Run Log**), generate a summary output file for the case (**Generate Case Summary**) and view the log file for optimization cases (**View Optimization Log**).

3.11.1 Viewing Electrostatic Results

The **Electrostatics** menu item will be active after completion of an electrostatic analysis. The first time you select this menu item following BOA execution, BoaGui will display the informational message shown in Figure 71. This message indicates that the Electrostatic Field Postprocessor must be run in order to generate the files necessary for display. Click ‘Yes’ and wait for the postprocessor to complete.


boaGui

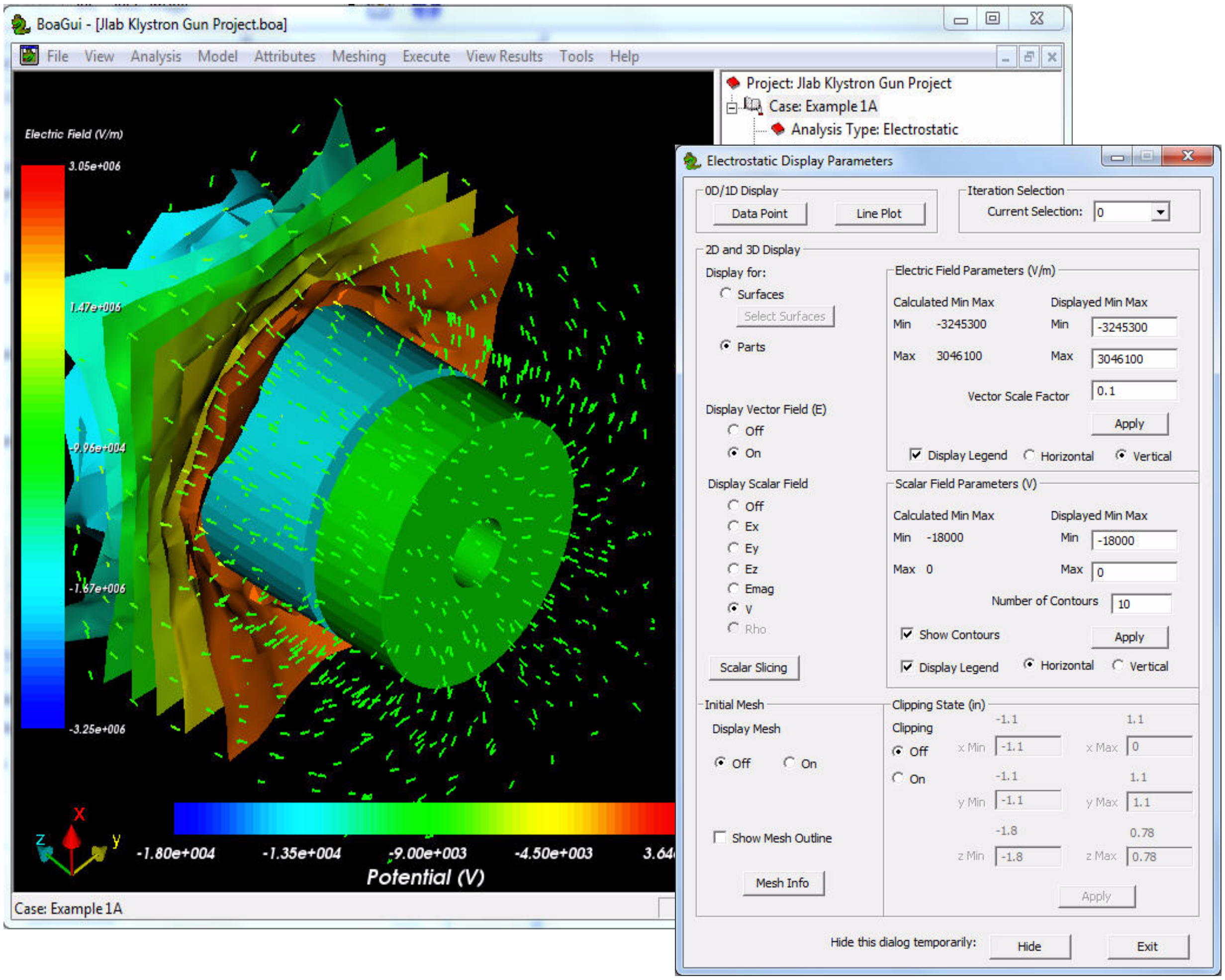
The first field display requires generation of the Electrostatic Field Postprocessor. This will take some time. Do you want to continue?

☐ Do not ask this question again

Yes
No

FIGURE 71. Postprocessor Informational Message

The default electrostatic results will appear in the main window and the *Electrostatic Display Parameters* dialog will automatically pop up on top as shown Figure 72. The displayed results correspond to the default settings on the *Electrostatic Display Parameters* dialog: *Display Vector Field (E)* set to ‘On’ and *Display Scalar Field* set to ‘V’ (i.e., display the voltage contours).


BoaGui - [Jlab Klystron Gun Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
Electric Field (V/m)
3.05e+006
1.47e+006
-9.96e+004
-1.67e+006
-3.25e+006
x
y
z
-1.80e+004
-1.35e+004
-9.00e+003
-4.50e+003
3.64
Potential (V)
Case: Example 1A
Electrostatic Display Parameters
0D/1D Display
Data Point
Line Plot
Select Surfaces
Apply
Scalar Slicing
Mesh Info
Apply
Hide
Exit

Iteration Selection
Current Selection: 0

2D and 3D Display
Display for:
○ Surfaces
● Parts

Display Vector Field (E)
○ Off
● On

Display Scalar Field
○ Off
○ Ex
○ Ey
○ Ez
○ Emag
● V
○ Rho

Electric Field Parameters (V/m)
Calculated Min Max: Min -3245300, Max 3046100
Displayed Min Max: Min -3245300, Max 3046100
Vector Scale Factor 0.1
☑ Display Legend ○ Horizontal ● Vertical

Scalar Field Parameters (V)
Calculated Min Max: Min -18000, Max 0
Displayed Min Max: Min -18000, Max 0
Number of Contours 10
☑ Show Contours
☑ Display Legend ● Horizontal ○ Vertical

Initial Mesh
Display Mesh ● Off ○ On
☐ Show Mesh Outline

Clipping State (in)
Clipping
● Off
○ On
-1.1 x Min -1.1 — 1.1 x Max 0
-1.1 y Min -1.1 — 1.1 y Max 1.1
-1.8 z Min -1.8 — 0.78 z Max 0.78

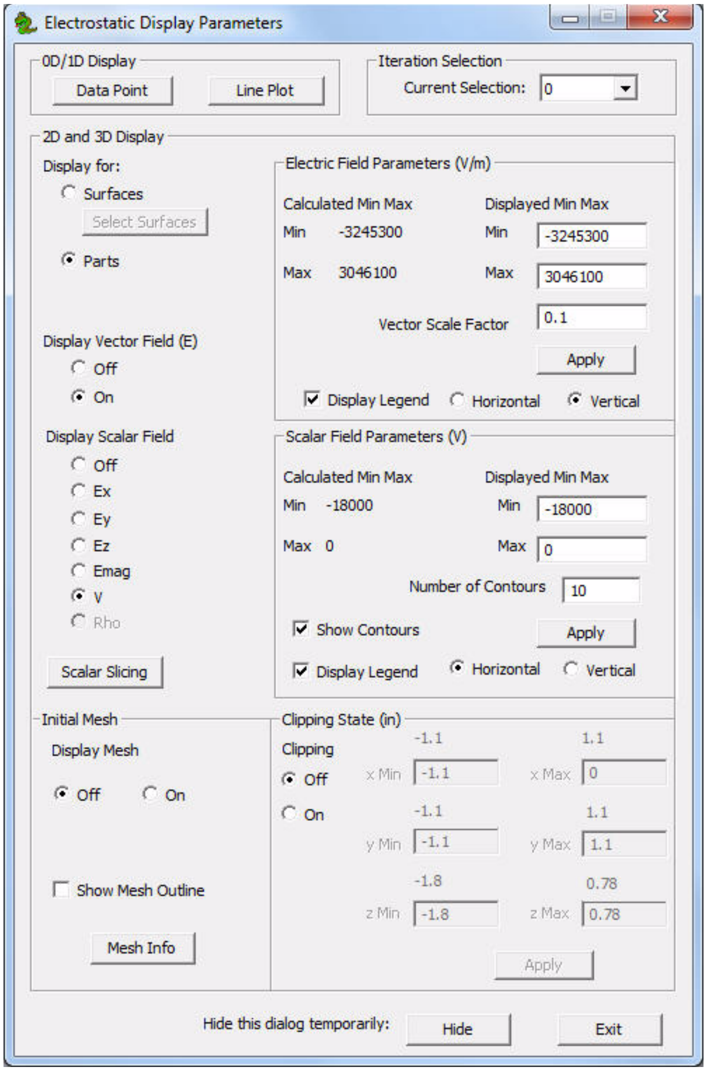
Hide this dialog temporarily:

FIGURE 72. Electrostatic results (default settings)

This *Electrostatic Display Parameters* dialog is shown more clearly in Figure 73. It includes the following sections:

- *Iteration Selection* - For beam analyses, the dropdown in the upper right corner of the dialog specifies the iteration for which to display the Electrostatic results. The contents of the dropdown box correspond to the saved iterations of the last run (as defined by the parameter *Output Solution and Source Fields Every {N} Iterations* on the *Execute* dialogs). The default iteration is the final iteration.
- *0D/1D Display* - The upper left portion of the dialog provides buttons for accessing the *Single Point Electrostatic Field Extractor* dialog (via the *Data Point* button) and the *Electrostatic Line Plotter* dialog (via the *Line Plot* button).

- *2D/3D Display* - The majority of the dialog is devoted to the controls for 2D and 3D display of the electrostatic results. This section is discussed next.



Electrostatic Display Parameters

0D/1D Display

Iteration Selection
Current Selection: 0

2D and 3D Display

Display for:
○ Surfaces

● Parts

Display Vector Field (E)
○ Off
● On

Display Scalar Field
○ Off
○ Ex
○ Ey
○ Ez
○ Emag
● V
○ Rho

Electric Field Parameters (V/m)

Calculated Min Max		Displayed Min Max	
Min	-3245300	Min	-3245300
Max	3046100	Max	3046100

Vector Scale Factor 0.1

☑ Display Legend ○ Horizontal ● Vertical

Scalar Field Parameters (V)

Calculated Min Max		Displayed Min Max	
Min	-18000	Min	-18000
Max	0	Max	0

Number of Contours 10
☑ Show Contours

☑ Display Legend ● Horizontal ○ Vertical

Initial Mesh
Display Mesh
● Off ○ On
☐ Show Mesh Outline

Clipping State (in)
Clipping
● Off
○ On

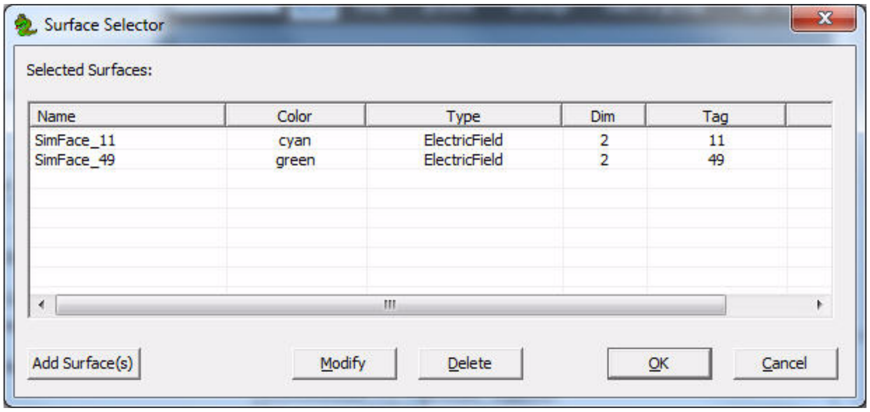
	-1.1		1.1
x Min	-1.1	x Max	0
	-1.1		1.1
y Min	-1.1	y Max	1.1
	-1.8		0.78
z Min	-1.8	z Max	0.78

Hide this dialog temporarily:

FIGURE 73. Electrostatic Display Parameters dialog

3.11.1.1 2D/3D Display

The 2D/3D section of the *Electrostatic Display Parameters* dialog provides controls for configuring the 2D and 3D display of the electrostatic results. It contains the following controls:



Selected Surfaces:

Name	Color	Type	Dim	Tag
SimFace_11	cyan	ElectricField	2	11
SimFace_49	green	ElectricField	2	49

FIGURE 74. Surface Selector dialog

The *Surface/Part Display* radio buttons control whether the results are shown for the whole geometry ('Parts') or selected Surfaces ('Surfaces'). The radio buttons initially default to 'Parts'. To view results for specific surfaces instead, change the radio button to 'Surfaces'. The dialog shown in Figure 74 will pop up. Initially the table will be blank. To specify a surface, click the *Add Surface* radio button. This will bring up the main window containing the geometric model. Find the desired surface and select it by pressing Ctrl-Left mouse button. Click *Enter* to select. *{TBD: Integrate email response 3/19/10; resolve open issues; also note that when the surface does not have a CAD name, by convention it will be named in the GUI as SimFace_xx where xx is its tag.}*Click *OK* to close the *Surface Selector* dialog.

The *Display Vector Field* radio buttons turn the electric vector field display on and off. Figure 75 shows an example of *Display Vector Field* set to 'On'.

The *Electric Field Parameters* section controls the appearance of the electric vector field display. This section is inactive if the *Display Vector Field* radio button is set to 'Off'. The parameter settings include the size of the vector arrows (*Vector Scale Factor*) and the *Displayed Min* and *Max* values for the field. The *Displayed Min* and *Max* values initially default to the *Calculated Min* and *Max* values. The *Calculated Min* and *Max* values appear for reference (this makes it easy to reset the values). The *Display Legend* checkbox turns the legend on and off and the radio buttons labeled *Horizontal* and *Vertical* dictate the legend placement.

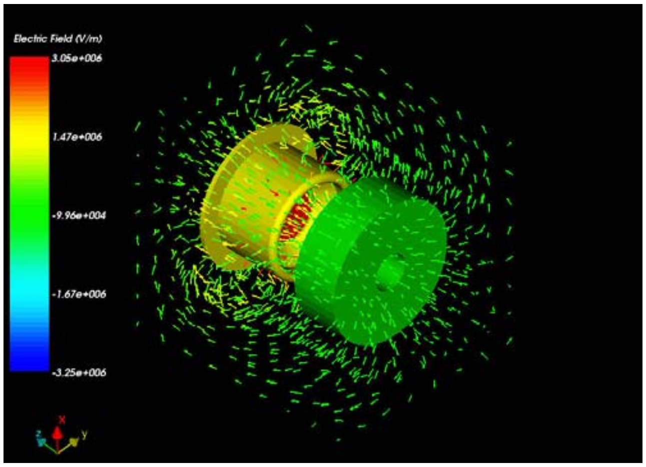


FIGURE 75. Display of Electric Vector Field

The *Display Scalar Field* radio buttons turn the display of the various scalar fields on and off. Options include:

- 'Off' (does not display any scalar fields)
- 'Ex' (displays magnitude of the x component of the electric field)
- 'Ey' (displays magnitude of the y component of the electric field)
- 'Ez' (displays magnitude of the z component of the electric field)

- ‘Emag’ (displays magnitude of the total electric field)
- ‘V’ (displays voltage)
- ‘Rho’ (displays charge density) (applies to electrostatic beam analyses only)

The *Scalar Field Parameters* section is active when the *Display Scalar Field* radio buttons are set to something other than ‘Off’. The *Show Contour* checkbox controls the appearance of the scalar field contour plot. If the checkbox is unchecked, the scalar field appears as a standard continuous contour plot like the ones shown in Figure 76. (The left-hand figure shows the contour plot for the outside of the meshed problem space. The right-hand figure uses extent clipping to reveal the internal structure. Extent clipping is discussed in a subsequent section.) Alternatively, if the *Show Contour* checkbox is checked, you specify *Number of Contours* and the display will group similar values into the specified number of discrete iso-surfaces as shown in Figure 77.

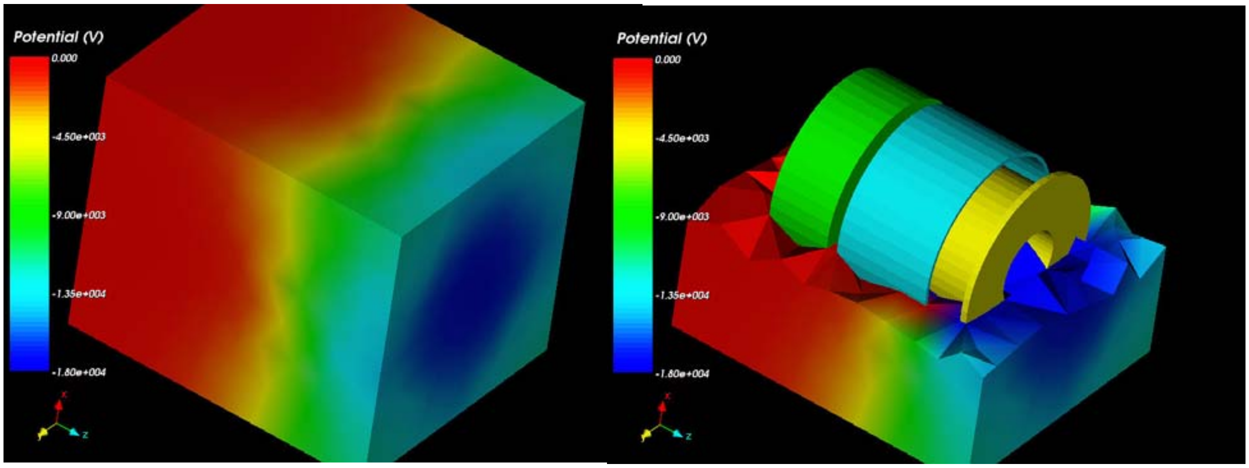


FIGURE 76. Voltage field contour plots, with the *Show Contours* checkbox unchecked (Right-hand figure uses extent clipping) CHANGE TO MATCH ORIENTATION BELOW

The *Displayed Min* and *Max* values dictate the upper and lower values displayed. These values initially default to the *Calculated Min* and *Max* values. (The *Calculated Min* and *Max* values appear for reference, making it easy to reset the values.) The *Display Legend* checkbox turns the legend display on and off, and the radio buttons labeled *Horizontal* and *Vertical* dictate the legend placement.

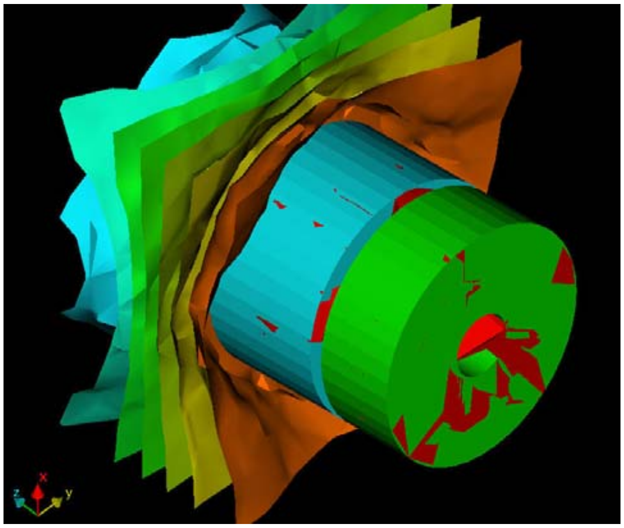
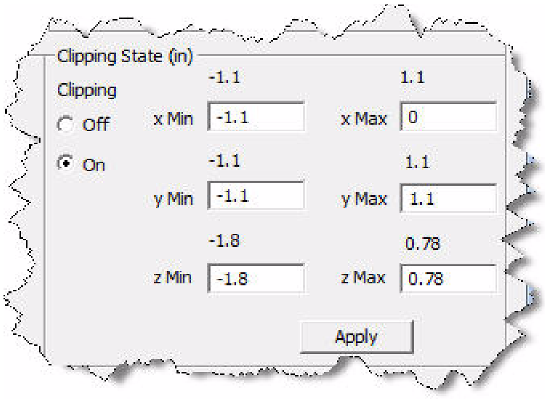


FIGURE 77. Voltage field contour plot, with *Show Contours* checked (i.e., iso - surfaces of voltage)

Initial (or Adapted) Mesh - This section controls the display of the mesh. It refers to the last mesh used in generating the results, either *Initial* (if Adaptivity is off) or *Adapted* (if adaptivity is on). The *Display Mesh* radio button shows/hides the mesh. The *Show Mesh Outline* checkbox displays an outline of the mesh. Figure 79 shows an example of the Adapted

Mesh turned On, the scalar field set to voltage and extent clipping used to remove a portion of the adapted mesh.

The *Clipping State* section, shown in Figure 78, allows you to remove sections of the displayed results in order to view the internal structure. When the *Clipping* radio button is set to ‘On’, the XYZ ranges (*x Min*, *x Max*, *y Min*, *y Max*, *z Min*, *z Max*) are used to determine which portion of the results to display. The ranges initially default to the full range of the problem space. For reference, these default values appear above each edit box (this makes it easy to reset the values). The units for all values (shown in parentheses after the words “Clipping state”) are the models units. Changes to the ranges do not take effect in the graphics window until you click the *Apply* button (or hit *Enter*).



Clipping State (in)

Clipping

○ Off

● On

x Min: -1.1 (default -1.1)

x Max: 0 (default 1.1)

y Min: -1.1 (default -1.1)

y Max: 1.1 (default 1.1)

z Min: -1.8 (default -1.8)

z Max: 0.78 (default 0.78)

FIGURE 78. Clipping State section of the Electrostatic Display Parameters dialog

Figure 79 shows two examples of extent clipping. In both, the clipping range corresponds to the values shown in Figure 78. In the left-hand figure, the electric vector field is ‘On’ and the scalar field is set to ‘voltage’ with *Show Contours* checked. In the right-hand figure the scalar field is set to ‘voltage’ with *Show Contours* unchecked and the mesh display is set to ‘On’.

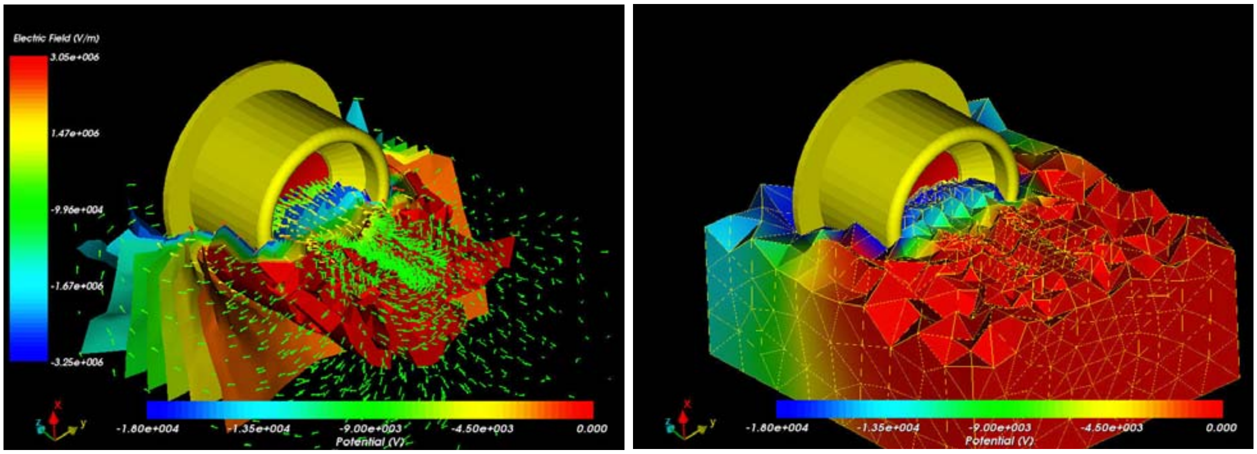
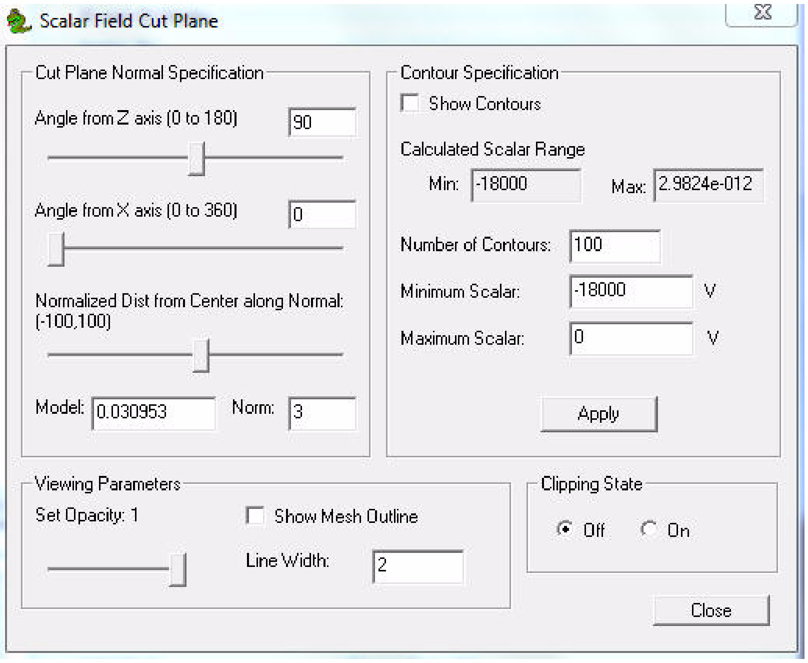


FIGURE 79. Extent clipping for two different result displays. Left-hand figure shows Voltage On (with Show Contours checked) and Vector Electric Fields On. Right-hand Figure shows Voltage On (with Show Contours unchecked) and Mesh On. Extent Clipping is set as shown in Figure 78.

The *Scalar Slicing* button (located beneath the *Scalar Field* radio buttons) accesses the *Scalar Field Cut Plane* dialog. The dialog allows you to define a plane anywhere within the problem boundary and view the scalar field distribution on that plane. (The scalar field shown corresponds to the selected radio button on the *Electrostatic Display Parameter* dialog.) The *Scalar Field Cut Plane* dialog is divided into the following sections:



Scalar Field Cut Plane

Cut Plane Normal Specification

Angle from Z axis (0 to 180): 90

Angle from X axis (0 to 360): 0

Normalized Dist from Center along Normal: (-100,100)

Model: 0.030953

Norm: 3

Contour Specification

☐ Show Contours

Calculated Scalar Range

Min: -18000

Max: 2.9824e-012

Number of Contours: 100

Minimum Scalar: -18000 V

Maximum Scalar: 0 V

Viewing Parameters

Set Opacity: 1

☐ Show Mesh Outline

Line Width: 2

Clipping State

● Off ○ On

FIGURE 80. Scalar Field Cut Plane dialog

Cut Plane Normal Specification - Use the slider bars to specify the exact position of the cut plane. Controls include: *Angle from the Z-axis* (from 0 to 180 degrees), *Angle from the X-axis* (from 0 to 360 degrees) and *Normalized Distance from Center Along Normal* (slides the plane along the normal to the surface). You can also enter specific values for these parameters in the edit boxes provided rather than use the slider bars. For *Normalized Distance from Center Along Normal*, you can enter in the distance in the model units (*Model*) or normalized units (*Norm*).

Contour Specifications - This section controls the display of the scalar field on the cut plane. Use the *Show Contours* checkbox to toggle the display between contour lines (checked) and a continuous color contour plot (unchecked). When viewing contour lines, define the contour range by specifying the *Number of Contours* to be displayed and the *Minimum Voltage* and *Maximum Voltage* that define the range. Note that changes do not take effect in the graphics display until you click the *Apply* button (or the *Enter* key).

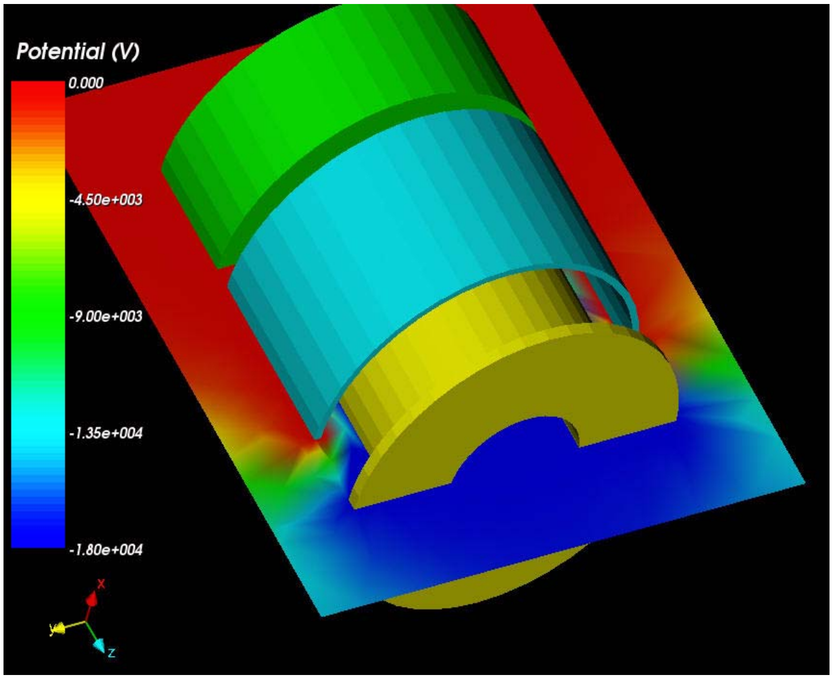


FIGURE 81. Scalar Field Cut Plane

Viewing Parameters - The *Set Opacity* slider bar sets the opacity of the scalar cut plane. To hide the cut plane, set the slider bar to zero. The *Line Width* sets the thickness of the contour lines. It applies only when the *Show Contours* checkbox is checked.

Clipping State - This radio button turns the clipping state on and off. For an explanation of Clipping State, see discussion of the Clipping State section of the *Electrostatic Display Parameters* dialog in the section below.)

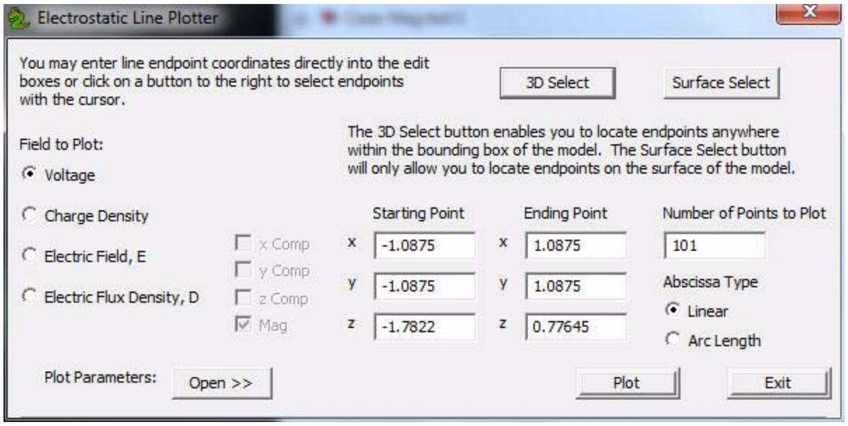
The *Close* button closes the *Scalar Field Cut Plane* dialog and returns to the *Electrostatic Display Parameters* dialog, shown in Figure 73.

{TBD: discuss *Hide* vs. *Exit;* Also do more bug-checking to see if I can reproduce the odd behavior I initially saw when using *Hide* on the jlab klystron case.}

3.11.1.2 01/1D Display (Line Plots & Data Point Extraction)

3.11.1.2.1 Line Plots

The *Line Plot* button in the *0D/1D* section of the *Electrostatic Display Parameters* dialog (shown in Figure 73) brings up the *Electrostatic Line Plotter* dialog shown in shown in Figure 82. This dialog enables you to create line plots for various fields. Use the *Field to Plot* radio buttons to select the desired field:



Electrostatic Line Plotter

You may enter line endpoint coordinates directly into the edit boxes or click on a button to the right to select endpoints with the cursor.

The 3D Select button enables you to locate endpoints anywhere within the bounding box of the model. The Surface Select button will only allow you to locate endpoints on the surface of the model.

Field to Plot:
- ● Voltage
- ○ Charge Density
- ○ Electric Field, E
- ○ Electric Flux Density, D

☐ x Comp
☐ y Comp
☐ z Comp
☑ Mag

	Starting Point	Ending Point
x	-1.0875	1.0875
y	-1.0875	1.0875
z	-1.7822	0.77645

Number of Points to Plot: 101

Abscissa Type
- ● Linear
- ○ Arc Length

FIGURE 82. Electrostatic Line Plotter dialog

- ‘Voltage’
- ‘Charge Density’ (applicable for electrostatic beam analyses only)
- ‘Electric field, E’ - Use the checkboxes to select ‘x Comp’, ‘y Comp’, ‘z Comp’ or ‘Mag’ (the x component, y component, z component or total magnitude of E, respectively)
- ‘Electric Flux Density, D’ - Use the checkboxes to select ‘x Comp’, ‘y Comp’, ‘z Comp’ or ‘Mag’ (the x component, y component, z component or total magnitude of D, respectively)

To specify the desired line for which to plot the selected field, you must identify a *Starting Point* and an *Ending point*. The dialog provides several options for identifying these two points:

- Direct entry - Allows you to enter the *x*, *y*, and *z* components of each point directly in the edit fields provided on the *Electrostatic Line Plotter* dialog (e.g., *Starting Point x*, *Starting Point* y etc.)

- Surface Selection - Allows you to select two points anywhere on the surface of the model. Click the *Surface Select* button and the main window containing the geometry model will become active. Use the Ctrl-Left mouse button to select the first and second end points. Only points on surfaces can be selected. A small dot will appear for each endpoint and the xyz coordinates will automatically appear in the fields provided on the *Electrostatic Line Plotter* dialog (e.g., *Starting Point x*, *Starting Point* y etc.).
- 3D Selection - Allows you to select two endpoints anywhere within the bounding box of the problem space. Click the *3D Select* button and the main window containing the geometry model will become active. A line widget with two endpoints will appear. Use the Ctrl-Left mouse button to move the endpoints to their desired locations. To complete the selection, click *3D Select* on the *Electrostatic Line Plotter* dialog. The xyz coordinates will automatically appear in the fields provided on the *Electrostatic Line Plotter* dialog (e.g., *Starting Point x*, *Starting Point* y etc.).

Once you specify the desired field and the desired line, click the *Plot* button. The resulting line plot will appear in the main window, as shown in Figure 83. In this case, the line plot corresponds to the settings shown in Figure 82.

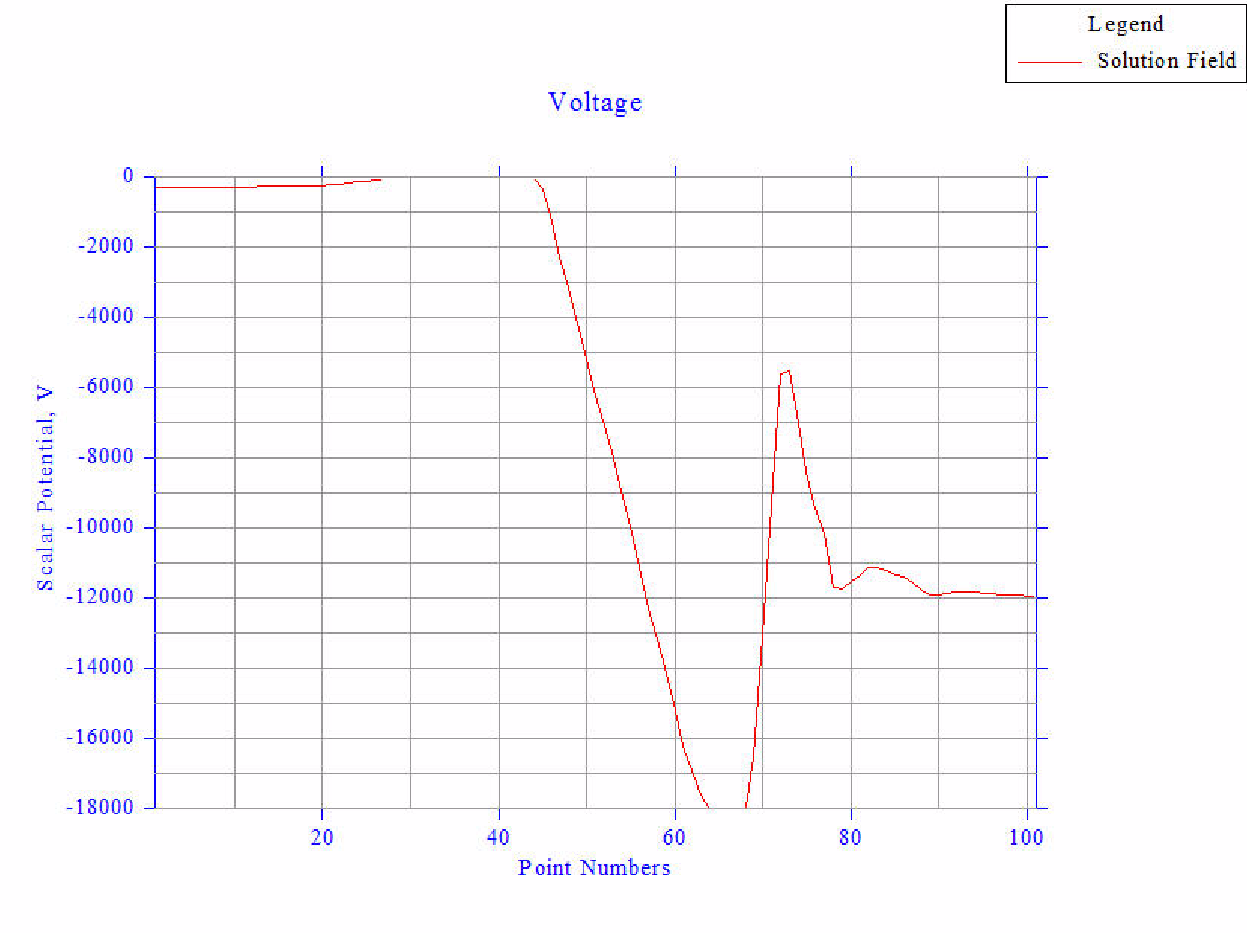
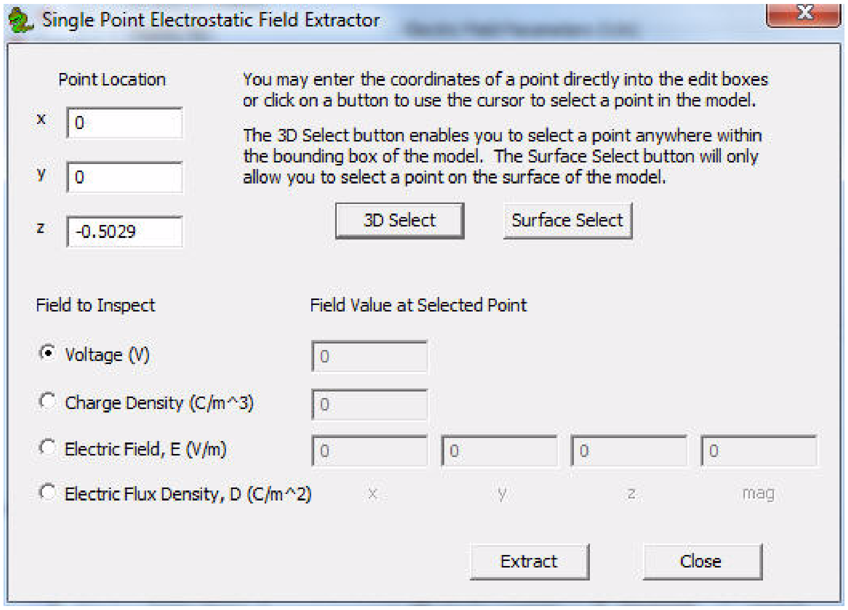


FIGURE 83. Voltage Line Plot

The *Plot Parameters Open* button opens a hidden portion at the bottom of the *Electrostatic Line Plotter* dialog. This section {TBD: discuss here and/or reference back to the earlier *Plot Parameters* section.}

To exit the *Electrostatic Line Plotter* dialog, click the exit button and control will be returned to the *Electrostatic Display Parameters* dialog.

3.11.1.2.2 Data Point Extraction



Single Point Electrostatic Field Extractor

Point Location

x 0

y 0

z -0.5029

You may enter the coordinates of a point directly into the edit boxes or click on a button to use the cursor to select a point in the model.

The 3D Select button enables you to select a point anywhere within the bounding box of the model. The Surface Select button will only allow you to select a point on the surface of the model.

Field to Inspect

Field Value at Selected Point

● Voltage (V) 0

○ Charge Density (C/m^3) 0

○ Electric Field, E (V/m) 0 0 0 0

○ Electric Flux Density, D (C/m^2)

FIGURE 84. Single Point Electrostatic Field Extractor

The *Data Point* button in the *0D/1D* section of the *Electrostatic Display Parameters* dialog (shown in Figure 73) brings up the *Single Point Electrostatic Field Extractor* dialog shown in Figure 84. This dialog enables you to extract the value of the desired field at a particular point. Use the *Field to Inspect* radio buttons to select the desired field:

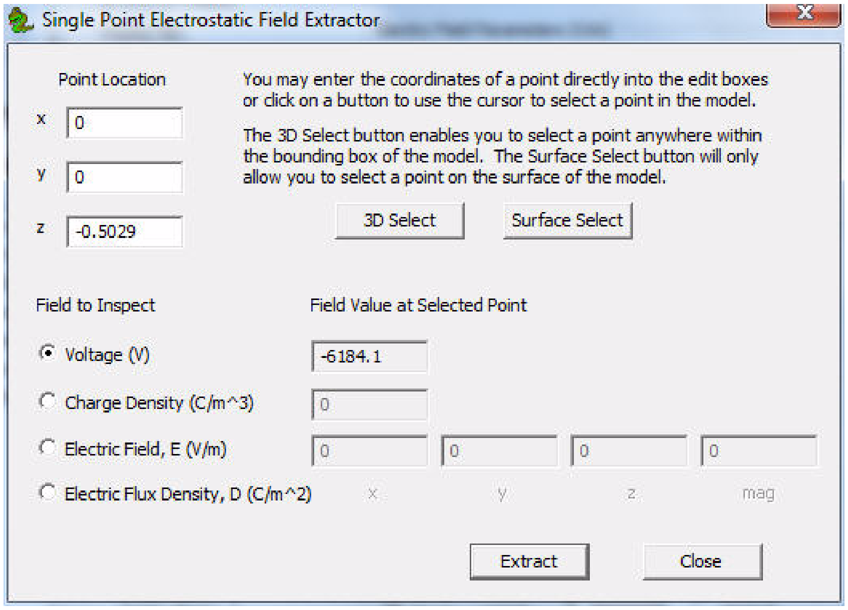
- ‘Voltage’
- ‘Charge Density’ (applicable for electrostatic beam analyses only)
- ‘Electric field, E’
- ‘Electric Flux Density, D’

There are three different ways to specify the desired point:

- Direct entry - Allows you to enter the *x*, *y*, and *z* components of the point directly into the edit fields provided on the *Single Point Electrostatic Field Extractor* dialog (*Point Location x*, *y* & *z*).
- Surface Selection - Allows you to select the point anywhere on the surface of the model. Click the *Surface Select* button and the main window containing the geometry model will become active. Use the Ctrl-Left mouse button to select the point. Only points on surfaces can be selected. A small dot will appear for the selected point and the xyz coordinates will automatically appear in the fields provided on the *Single Point Electrostatic Field Extractor* dialog (*Point Location x*, *y* & *z*).
- 3D Selection - Allows you to select the point anywhere within the bounding box of the problem space. Click the *3D Select* button and the main window containing the geometry model will become active. A 3-plane widget will appear. Use the Ctrl-Left mouse button to move the planes to their desired positions. To complete the selection, click *3D Select* on the *Single Point*

Electrostatic Field Extractor dialog. The xyz coordinates of the point represented by the intersection of the three planes will automatically appear in the fields provided on the *Single Point Electrostatic Field Extractor* dialog (*Point Location x*, *y* & *z*).

Once you specify the desired field and the desired point, click the *Extract* button. The resulting value(s) will appear in the appropriate *Field Value at Selected Point* box(es), as shown in Figure 85. Note that when *Field to Inspect* is set to ‘Electric Field’ or ‘Electric Flux Density’, the *Field Value at Selected Point* includes values for the x, y and z components as well as the total magnitude of the field.



Single Point Electrostatic Field Extractor

Point Location

x 0

y 0

z -0.5029

You may enter the coordinates of a point directly into the edit boxes or click on a button to use the cursor to select a point in the model.

The 3D Select button enables you to select a point anywhere within the bounding box of the model. The Surface Select button will only allow you to select a point on the surface of the model.

Field to Inspect — Field Value at Selected Point

● Voltage (V) -6184.1

○ Charge Density (C/m^3) 0

○ Electric Field, E (V/m) 0 0 0 0

○ Electric Flux Density, D (C/m^2) x y z mag

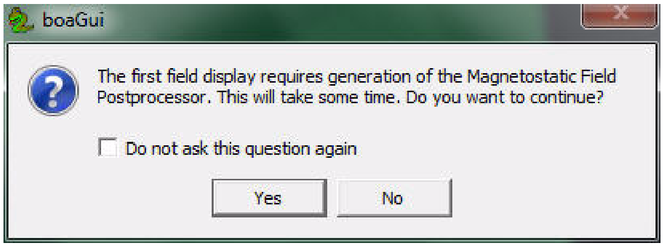
FIGURE 85. Results from Single Point Electrostatic Field Extraction

To exit the *Single Point Electrostatic Field Extractor* dialog, click the *Exit* button and control returns to the *Electrostatic Display Parameters* dialog.

3.11.2 Viewing Magnetostatic Results

{THIS SECTION IS INCOMPLETE. Complete after I get new Magnetostatic example from MR}

The **Magnetostatics** menu item will be active after completion of a magnetostatic analysis. The first time you select the *Magnetostatics* menu item following BOA execution, BoaGui will display the informational message shown in Figure 86. This message indicates that the Magnetostatic Field Postprocessor must be run in order to generate the files necessary for display. Click ‘Yes’ and wait for the postprocessor to complete.



boaGui

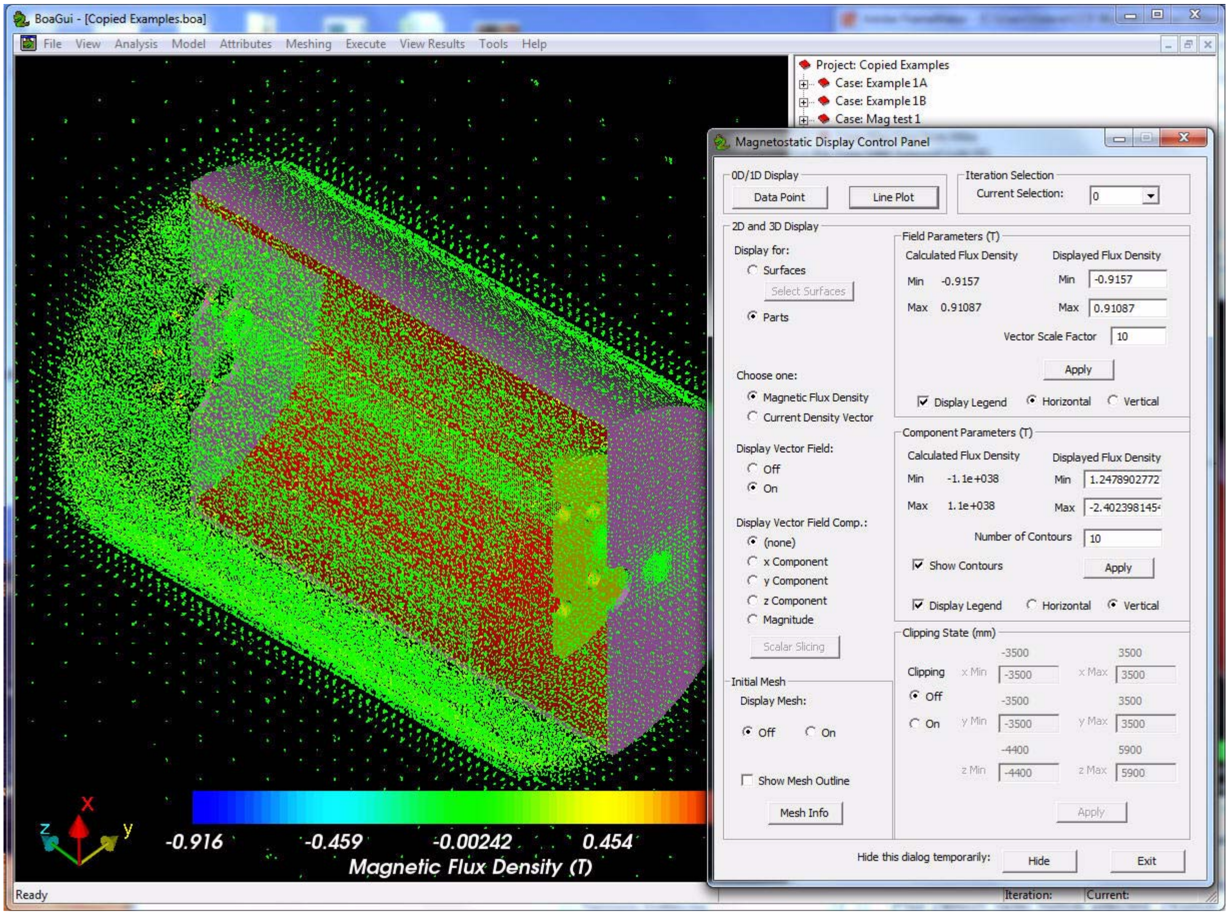
The first field display requires generation of the Magnetostatic Field Postprocessor. This will take some time. Do you want to continue?

☐ Do not ask this question again

FIGURE 86. Postprocessor Informational Message

{TBD - waiting for case to complete this section}

The default magnetostatic results will appear in the main window and the *Magnetostatic Display Parameters* dialog will automatically pop up on top as shown Figure 87. The displayed results correspond to the default settings on the *Magnetostatic Display Parameters* dialog: *Magnetostatic Flux Density* radio button selected, *Display Vector Field* set to 'On' and *Display Vector Field Component* set to 'none'.


BoaGui - [Copied Examples.boa]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
Project: Copied Examples
Case: Example 1A
Case: Example 1B
Case: Mag test 1
x
y
z
-0.916
-0.459
-0.00242
0.454
Magnetic Flux Density (T)
Ready
Iteration:
Current:

Magnetostatic Display Control Panel

0D/1D Display: Data Point | Line Plot

Iteration Selection — Current Selection: 0

2D and 3D Display

Display for:
○ Surfaces (Select Surfaces)
● Parts

Choose one:
● Magnetic Flux Density
○ Current Density Vector

Display Vector Field:
○ Off
● On

Display Vector Field Comp.:
● (none)
○ x Component
○ y Component
○ z Component
○ Magnitude
(Scalar Slicing)

Field Parameters (T)
Calculated Flux Density: Min -0.9157, Max 0.91087
Displayed Flux Density: Min -0.9157, Max 0.91087
Vector Scale Factor: 10
(Apply)
☑ Display Legend ● Horizontal ○ Vertical

Component Parameters (T)
Calculated Flux Density: Min -1.1e+038, Max 1.1e+038
Displayed Flux Density: Min 1.2478902772, Max -2.402398145
Number of Contours: 10
☑ Show Contours (Apply)
☑ Display Legend ○ Horizontal ● Vertical

Initial Mesh
Display Mesh: ● Off ○ On
☐ Show Mesh Outline
(Mesh Info)

Clipping State (mm)
Clipping: ● Off ○ On
x Min -3500 (-3500), x Max 3500 (3500)
y Min -3500 (-3500), y Max 3500 (3500)
z Min -4400 (-4400), z Max 5900 (5900)
(Apply)

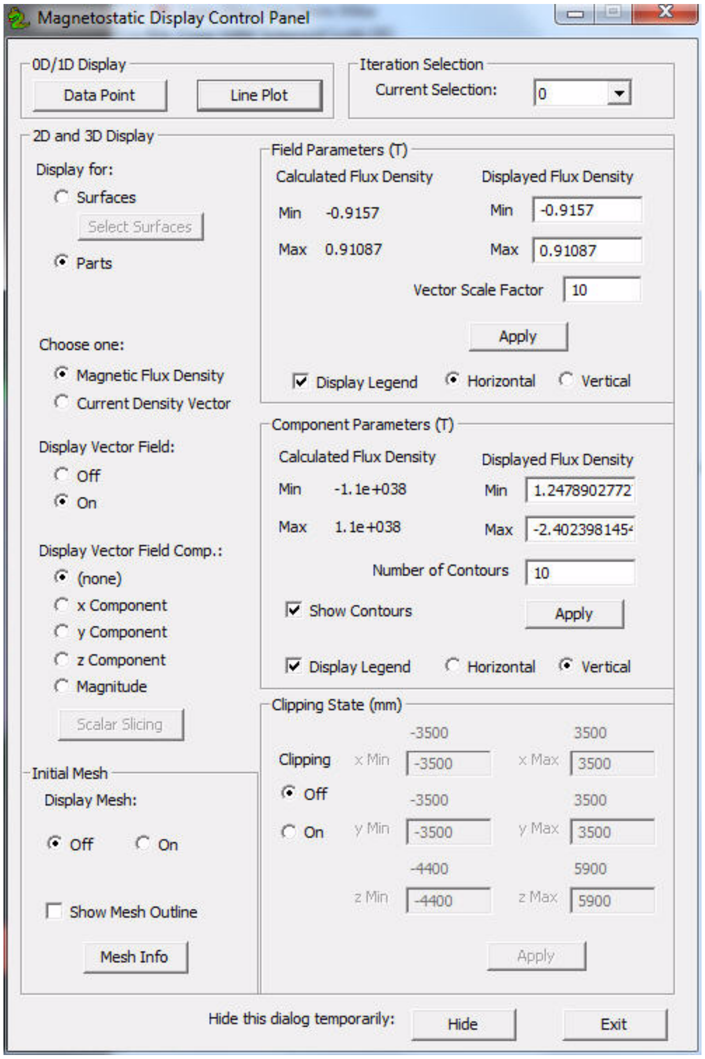
Hide this dialog temporarily: (Hide) (Exit)

FIGURE 87. Magnetostatic results (default settings) GET REAL CASE PICTURE

This *Magnetostatic Display Parameters* dialog is shown more clearly in Figure 88. It includes the following sections:

- {*Iteration Selection* - confirm this has been removed} IT HAS. UPDATE ALL PICS.
- *0D/1D Display* - The upper left portion of the dialog provides buttons for accessing the *Single Point Magnetostatic Field Extractor* dialog (via the *Data Point* button) and the *Magnetostatic Line Plotter* dialog (via the *Line Plot* button).

- *2D/3D Display* - The majority of the dialog is devoted to the controls for 2D and 3D display of the magnetostatic results. This section is discussed next.


Magnetostatic Display Control Panel
0D/1D Display
Data Point
Line Plot
Iteration Selection
2D and 3D Display
Field Parameters (T)
Apply
Component Parameters (T)
Apply
Clipping State (mm)
Apply
Select Surfaces
Scalar Slicing
Initial Mesh
Mesh Info
Hide
Exit

Current Selection: 0

Display for:
○ Surfaces
● Parts

Choose one:
● Magnetic Flux Density
○ Current Density Vector

Display Vector Field:
○ Off
● On

Display Vector Field Comp.:
● (none)
○ x Component
○ y Component
○ z Component
○ Magnitude

Display Mesh:
● Off ○ On

☐ Show Mesh Outline

Field Parameters (T)

Calculated Flux Density		Displayed Flux Density	
Min	-0.9157	Min	-0.9157
Max	0.91087	Max	0.91087

Vector Scale Factor 10

☑ Display Legend ● Horizontal ○ Vertical

Component Parameters (T)

Calculated Flux Density		Displayed Flux Density	
Min	-1.1e+038	Min	1.2478902772
Max	1.1e+038	Max	-2.402398145

Number of Contours 10

☑ Show Contours

☑ Display Legend ○ Horizontal ● Vertical

Clipping State (mm)

Clipping
● Off
○ On

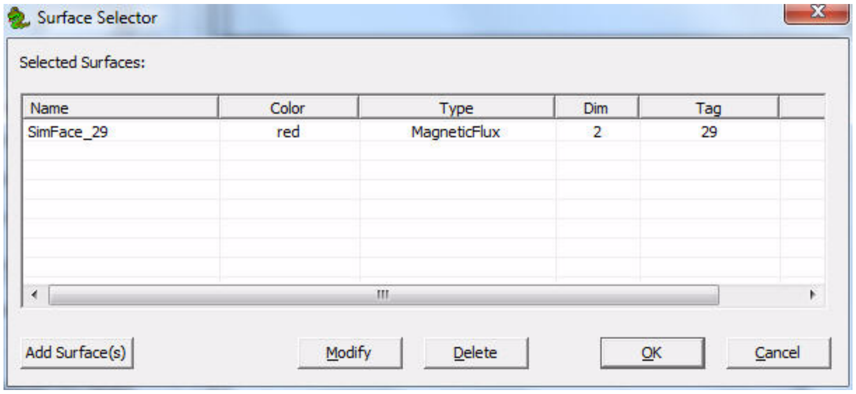
x Min	-3500	x Max	3500
y Min	-3500	y Max	3500
z Min	-4400	z Max	5900

Hide this dialog temporarily:

FIGURE 88. Magnetostatic Display Parameters dialog
(UPDATE: iteration selection gone & MAGNETIC FIELD

3.11.2.1 2D/3D Display

The 2D/3D section of the *Magnetostatic Display Parameters* dialog provides controls for configuring the 2D and 3D display of the magnetostatic results. It contains the following controls:



Selected Surfaces:

Name	Color	Type	Dim	Tag
SimFace_29	red	MagneticFlux	2	29

FIGURE 89. Surface Selector dialog

The *Surface/Part Display* radio buttons control whether the results are shown for the whole geometry (‘Parts’) or selected Surfaces (‘Surfaces’). The radio buttons initially default to ‘Part’. To view results for specific surfaces instead, change the radio button to ‘Surfaces’. The dialog shown in Figure 89 will pop up. Initially the table will be blank. To specify a surface, click the *Add Surface* radio button. This will bring up the main window containing the geometric model. Find the desired surface and select it by pressing Ctrl-Left mouse button. Click *Enter* to select. *{TBD: Integrate email response 3/19/10; resolve open issues; also note that when the surface does not have a CAD name, by convention it will be named in the GUI as SimFace_xx where xx is its tag.}*Click *OK* to close the *Surface Selector* dialog.

The *Vector Field Type* radio butttons control the type of vector field displayed:

- ‘Magnetic Flux Density’,
- ‘Current Density Vector’, or
- ‘Magnetic Field’.

The *Display Vector Field* radio buttons turn the selected vector field on and off. Figure 90 shows an example of *Display Vector Field* set to ‘On’ for *Vector Field Type* ‘Current Density Vector’.

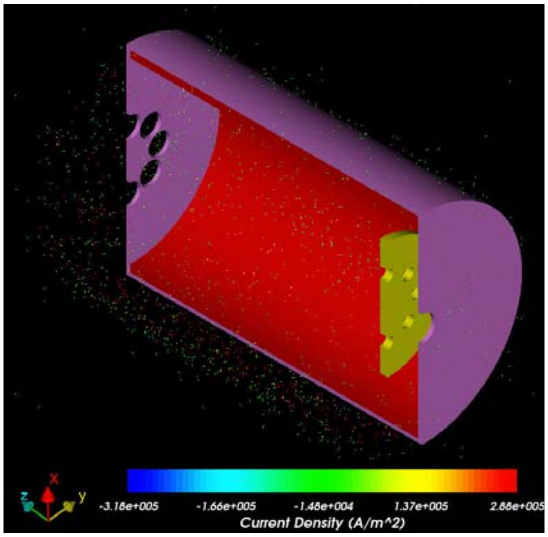


FIGURE 90. Display of Current Density

The *Field Parameters* section controls the appearance of the selected vector field. This section is inactive if the *Display Vector Field* radio button is set to ‘Off’. The parameter settings include the size of the vector arrows (*Vector Scale Factor*) and the *Displayed Min* and *Max* values for the field. The *Displayed Min* and *Max* values initially default to the *Calculated Min* and *Max* values. The *Calculated Min* and *Max* values appear for reference (this makes it easy to reset the values). The *Display Legend* checkbox turns the legend on and off and the radio buttons labeled *Horizontal* and *Vertical* dictate the legend location.

The *Display Vector Field Comp* radio buttons turn the display of the various vector field components on and off. Options include:

- ‘Off’ (does not display any components)
- ‘x Component’ (displays magnitude of the x component of the selected vector field)
- ‘y Component’ (displays magnitude of the y component of the selected vector field)
- ‘z Component’ (displays magnitude of the z component of the selected vector field)
- ‘Magnitude’ (displays total magnitude of the selected vector field)

The *Component Parameters* section is active when the *Display Vector Field Comp* radio buttons are set to something other than ‘Off’. The *Show Contour* checkbox controls the appearance of the contour plot. If the checkbox is unchecked, the field appears as a standard continuous contour plot like the one shown in Figure 91. (In this figure, extent clipping is used to reveal the internal structure. Extent clipping is discussed in a subsequent section.) Alternatively, if the *Show Contour* checkbox is checked, you specify *Number of Contours* and the display will group similar values into the specified number of discrete contour surfaces (i.e., iso-surfaces), as shown in Figure 92.

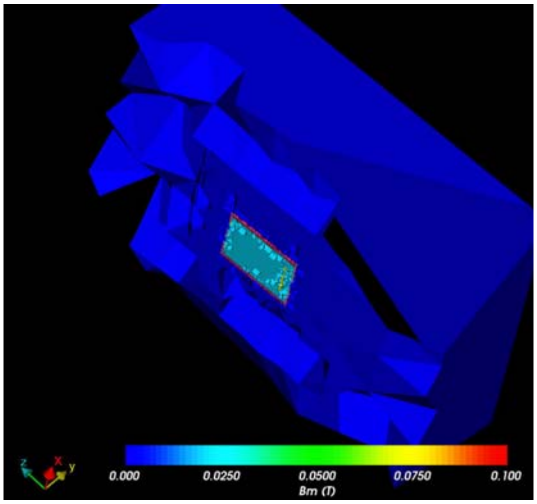


FIGURE 91. Contour plot for Magnitude of Magnetic Flux Density, with *Show Contours* unchecked

The *Displayed Min* and *Max* values dictate the upper and lower values displayed. These values initially default to the *Calculated Min* and *Max* values. (The *Calculated Min* and *Max* values appear for reference, making it easy to reset the values.) The *Display Legend* checkbox turns the legend display on and off, and the radio buttons labeled *Horizontal* and *Vertical* dictate the legend placement.

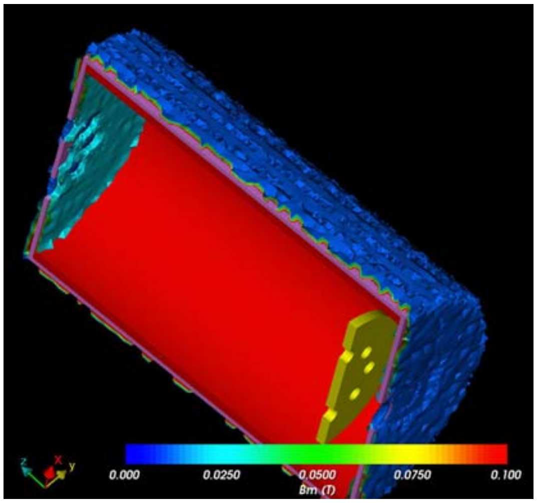
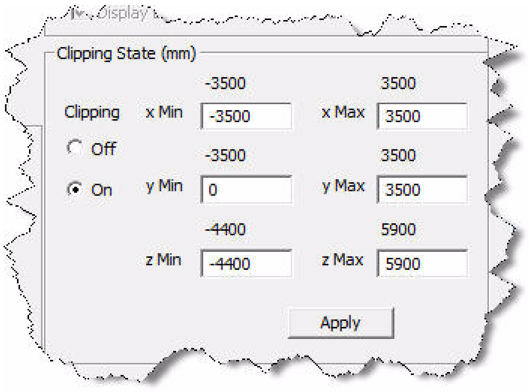


FIGURE 92. Contour plot for Magnitude of Magnetic Flux Density, with *Show Contours* checked (10 iso-surfaces)

The *Initial (or Adapted) Mesh* section controls the display of the mesh. It refers to the last mesh used in the generation of the results, either Initial (if adaptivity is off) or Adapted (if adaptivity is on) in the graphics window. Use the *Display Mesh* radio button to show or hide the mesh. The *Show Mesh Outline* checkbox to display an outline of the mesh.

The *Clipping State* section allows you to remove sections of the displayed results in order to view the internal structure. When the *Clipping* radio button is set to ‘On’, the XYZ ranges (*x Min*, *x Max*, *y Min*, *y Max*, *z Min*, *z Max*) are used to determine which portion of the results to display. The ranges initially default to the full range of the problem space. For reference, these default values appear above each edit box (this makes it easy to reset the values). The units for all values (shown in parentheses after the words “Clipping State”) are the model units. Changes to the ranges do not take effect in the graphics window until you click the *Apply* button (or hit *Enter*).



Clipping State (mm)

Clipping: ○ Off ● On

x Min (default -3500): -3500

x Max (default 3500): 3500

y Min (default -3500): 0

y Max (default 3500): 3500

z Min (default -4400): -4400

z Max (default 5900): 5900

FIGURE 93. Clipping State section of the Magnetostatic Display Parameters dialog

(The Clipping feature operates identically for both the Magnetostatic and Electrostatic results. Refer to Figure 78 and Figure 79 (on page 68) in the Electrostatic results section to see additional examples of extent clipping.)

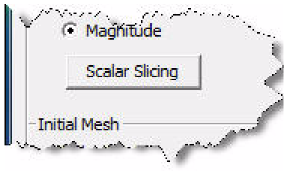
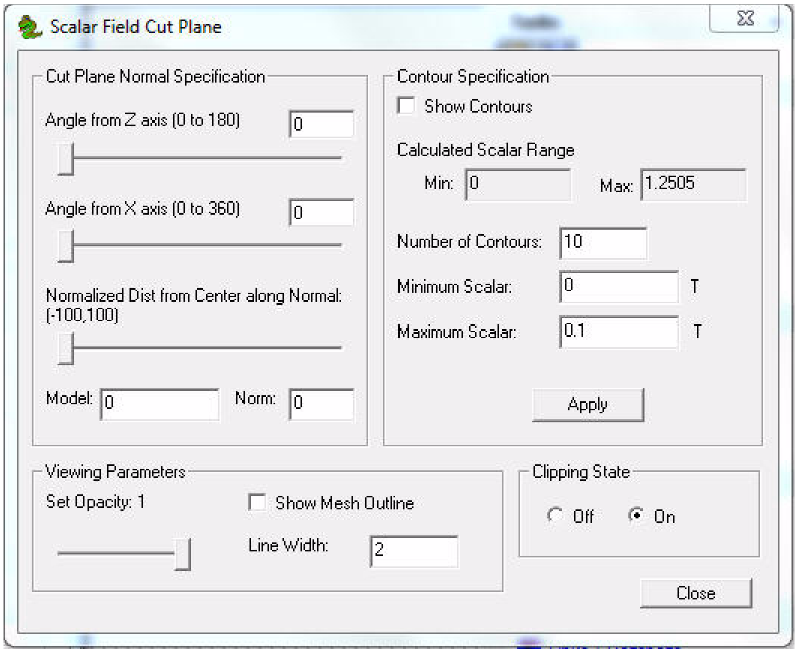


FIGURE 94. Scalar Slicing button

The *Scalar Slicing* button located beneath the *Display Vector Field Comp* radio buttons (see Figure 94) accesses the *Scalar Field Cut Plane* dialog shown in Figure 95. The dialog allows you to define a plane anywhere within the problem boundary and view the scalar field distribution on that plane. The scalar field shown corresponds to the selected vector field component radio button on the *Magnetostatic Display Parameter* dialog. The *Scalar Field Cut Plane* dialog is divided into the following sections:



Cut Plane Normal Specification

Angle from Z axis (0 to 180): 0

Angle from X axis (0 to 360): 0

Normalized Dist from Center along Normal: (-100,100)

Model: 0

Norm: 0

Contour Specification

☐ Show Contours

Calculated Scalar Range

Min: 0

Max: 1.2505

Number of Contours: 10

Minimum Scalar: 0 T

Maximum Scalar: 0.1 T

Viewing Parameters

Set Opacity: 1

☐ Show Mesh Outline

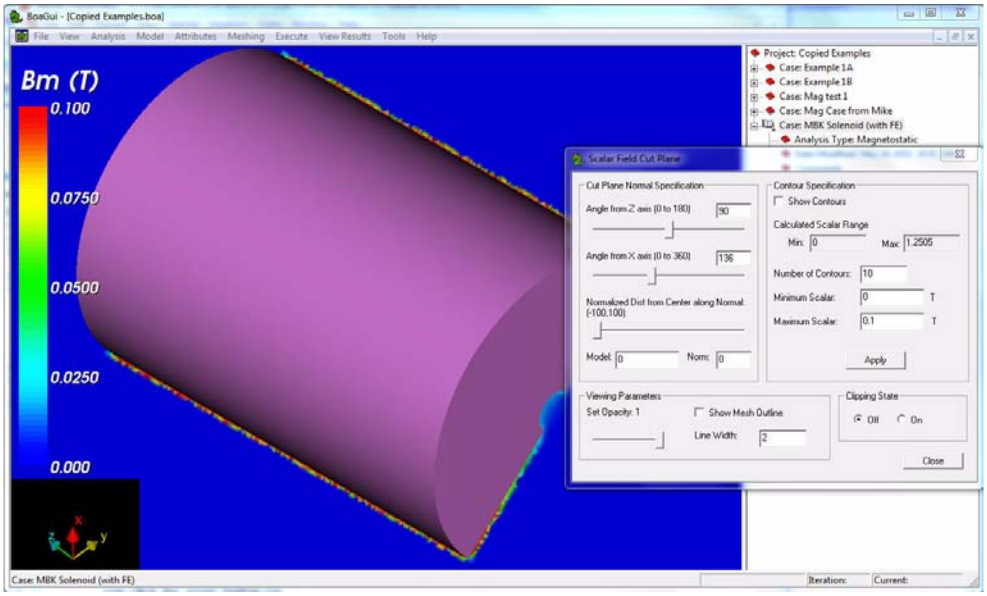
Line Width: 2

Clipping State

○ Off ● On

FIGURE 95. Scalar Field Cut Plane dialog

Cut Plane Normal Specification - Use the slider bars to specify the exact position of the cut plane. Controls include: *Angle from the Z-axis* (from 0 to 180 degrees), *Angle from the X-axis* (from 0 to 360 degrees) and *Normalized Distance from Center Along Normal* (slides the plane along the normal to the surface). You can also enter specific values for these parameters in the edit boxes provided rather than use the slider bars. For *Normalized Distance from Center Along Normal*, you can enter in the distance in the model units (*Model*) or normalized units (*Norm*).


BoaGui - [Copied Examples.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Bm (T)
0.100
0.0750
0.0500
0.0250
0.000
z
x
y
Project: Copied Examples
Case: Example 1A
Case: Example 1B
Case: Mag test 1
Case: Mag Case from Mike
Case: MBK Solenoid (with FE)
Analysis Type: Magnetostatic
Scalar Field Cut Plane
Cut Plane Normal Specification
Contour Specification
Calculated Scalar Range
Viewing Parameters
Clipping State
Apply
Close
Iteration:
Current:

Angle from Z axis (0 to 180): 90
Angle from X axis (0 to 360): 136
Normalized Dist from Center along Normal (-100,100)
Model: 0
Norm: 0
☐ Show Contours
Min: 0
Max: 1.2505
Number of Contours: 10
Minimum Scalar: 0 T
Maximum Scalar: 0.1 T
Set Opacity: 1
☐ Show Mesh Outline
Line Width: 2
● Off ○ On
Case: MBK Solenoid (with FE)

FIGURE 96. Scalar Field Cut Plane

Contour Specifications - This section controls the display of the scalar field on the cut plane. Use the *Show Contours* checkbox to toggle the display between contour lines (checked) and a continuous color contour plot (unchecked). When viewing contour lines, define the contour range by specifying the *Number of Contours* to be displayed and the *Minimum Voltage* and *Maximum Voltage* that define the range. Note that changes do not take effect in the graphics display until you click the *Apply* button (or the *Enter* key).

Viewing Parameters - The *Set Opacity* slider bar sets the opacity of the scalar cut plane. To hide the cut plane, set the slider bar to zero. The *Line Width* sets the thickness of the contour lines. It applies only when the *Show Contours* checkbox is checked.

Clipping State - This radio button turns the clipping state on and off. For an explanation of Clipping State, see discussion of the *Clipping State* section of the *Magnetostatic Display Parameters* dialog in the section below.)

The *Close* button closes the *Scalar Field Cut Plane* dialog and returns to the *Magnetostatic Display Parameters* dialog, shown in Figure 88.

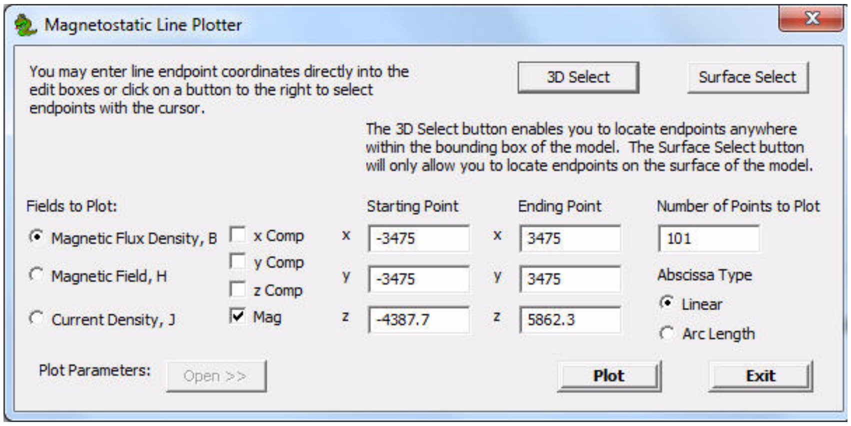
{Discuss *Hide* vs. *Exit* buttons}

3.11.2.2 01/1D Display (Line Plots & Data Point Extraction)

3.11.2.2.1 Line Plots

The *Line Plot* button in the *0D/1D* section of the *Magnetostatic Display Control Panel* dialog (shown in Figure 88) brings up the *Magnetostatic Line Plotter* dialog shown in Figure 97. This dialog enables you to create line plots for various fields. Use the *Field to Plot* radio buttons to select the desired field:

- ‘Magnetic Flux Density, B’
- ‘Magnetic Field, H’
- ‘Current Density, J’



You may enter line endpoint coordinates directly into the edit boxes or click on a button to the right to select endpoints with the cursor.

The 3D Select button enables you to locate endpoints anywhere within the bounding box of the model. The Surface Select button will only allow you to locate endpoints on the surface of the model.

Fields to Plot:
● Magnetic Flux Density, B
○ Magnetic Field, H
○ Current Density, J

☐ x Comp
☐ y Comp
☐ z Comp
☑ Mag

	Starting Point	Ending Point
x	-3475	3475
y	-3475	3475
z	-4387.7	5862.3

Number of Points to Plot: 101

Abscissa Type
● Linear
○ Arc Length

FIGURE 97. Magnetostatic Line Plotter dialog

Use the checkboxes to select ‘x Comp’, ‘y Comp’, ‘z Comp’ or ‘Mag’ (the x component, y component, z component or total magnitude of the selected field, respectively)

To specify the desired line for which to plot the selected field, you must identify a *Starting Point* and an *Ending point*. The dialog provides several options for identifying these two points:

- Direct entry - Allows you to enter the *x*, *y*, and *z* components of each point directly in the edit fields provided on the *Magnetostatic Line Plotter* dialog (e.g., *Starting Point x*, *Starting Point* y etc.)
- Surface Selection - Allows you to select two points anywhere on the surface of the model. Click the *Surface Select* button and the main window containing the geometry model will become active. Use the Ctrl-Left mouse button to select the first and second end points. Only points on surfaces can be selected. A small dot will appear for each endpoint and the xyz coordinates will automatically appear in the fields provided on the *Magnetostatic Line Plotter* dialog (e.g., *Starting Point x*, *Starting Point* y etc.).
- 3D Selection - Allows you to select two endpoints anywhere within the bounding box of the problem space. Click the *3D Select* button and the main window containing the geometry model will become active. A line widget with two endpoints will appear. Use the Ctrl-Left mouse button to move the endpoints to their desired locations. To complete the selection, click *3D Select* on the *Magnetostatic Line Plotter* dialog. The xyz coordinates will automatically appear in the fields provided on the *Magnetostatic Line Plotter* dialog (e.g., *Starting Point x*, *Starting Point* y etc.).

Once you specify the desired field and the desired line, click the *Plot* button. The resulting line plot will appear in the main window, as shown in Figure 98. In this case, the line plot corresponds to the settings shown in Figure 97.

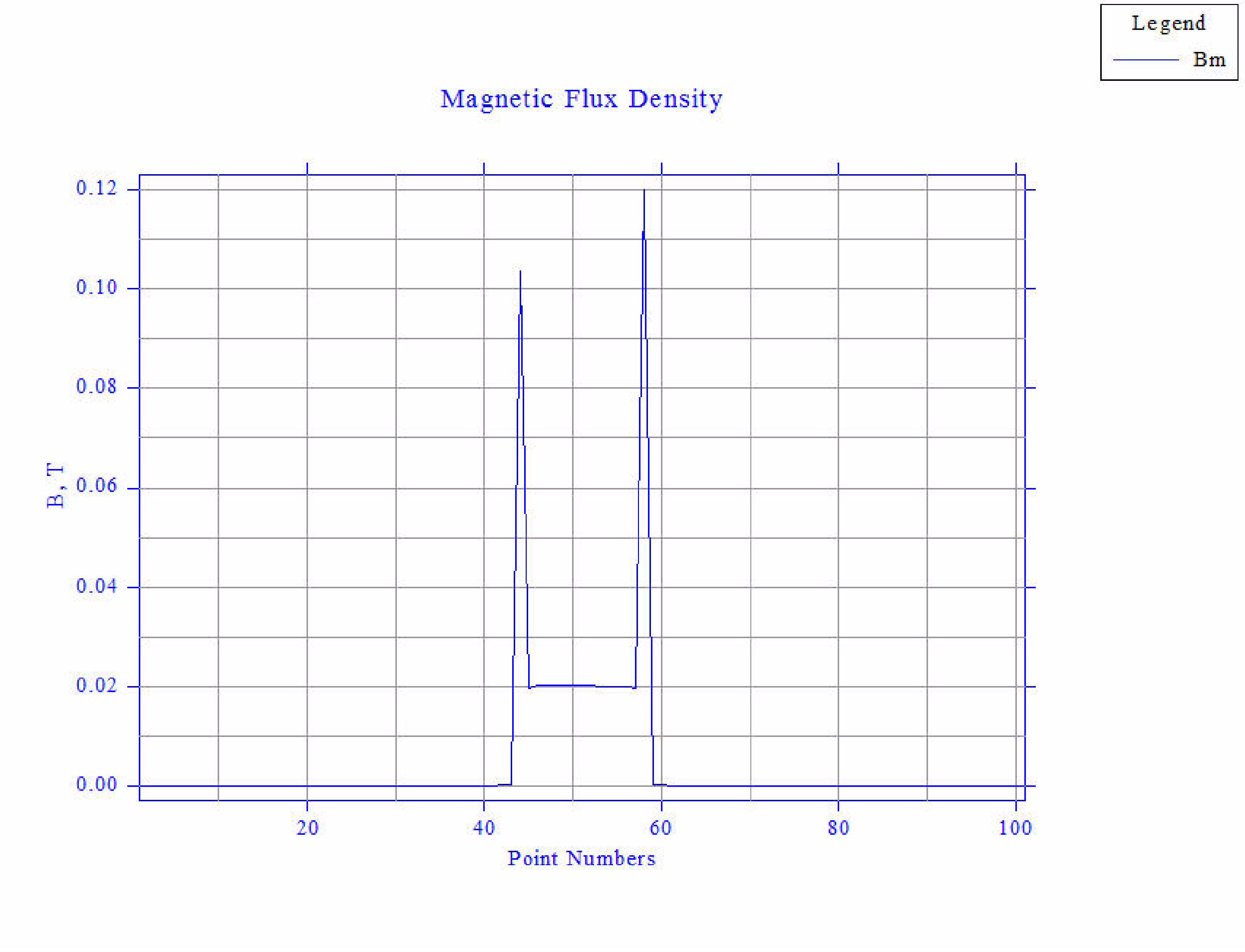


FIGURE 98. Magnetic Flux Density Line Plot

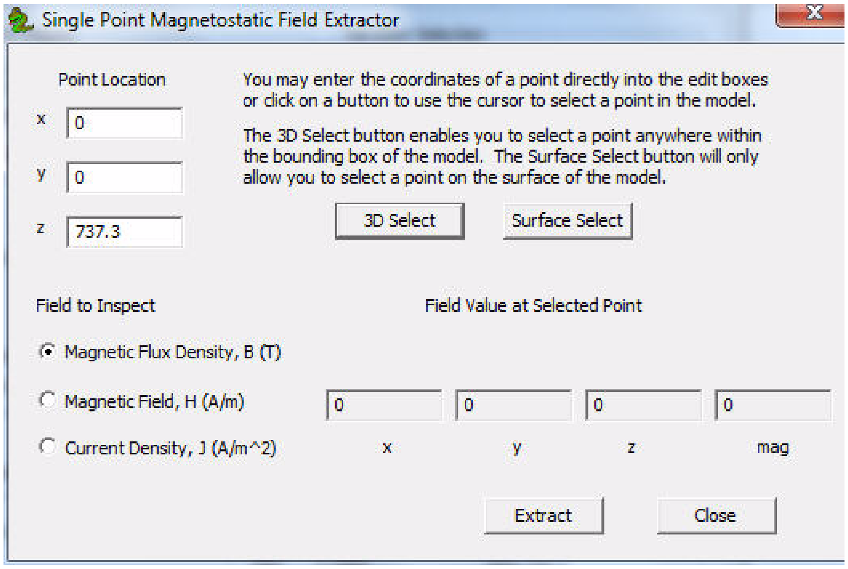
The *Plot Parameters Open* button opens a hidden portion at the bottom of the *Electrostatic Line Plotter* dialog. This section {TBD: discuss here and/or reference back to the earlier *Plot Parameters* section.}

To exit the *Magnetostatic Line Plotter* dialog, click the exit button and control will be returned to the *Magnetostatic Display Parameters* dialog.

3.11.2.2.2 Data Point Extraction

The *Data Point* button in the *0D/1D* section of the *Magnetostatic Display Parameters* dialog (shown in Figure 88) brings up the *Single Point Magnetostatic Field Extractor* dialog shown in Figure 99. This dialog enables you to extract the value of the desired field at a particular point. Use the *Field to Inspect* radio buttons to select the desired field:

- ‘Magnetic Flux Density, B’
- ‘Magnetic Field, H’
- ‘Current Density, J’


Single Point Magnetostatic Field Extractor

Point Location

x 0

y 0

z 737.3

You may enter the coordinates of a point directly into the edit boxes or click on a button to use the cursor to select a point in the model.

The 3D Select button enables you to select a point anywhere within the bounding box of the model. The Surface Select button will only allow you to select a point on the surface of the model.

Field to Inspect

● Magnetic Flux Density, B (T)

○ Magnetic Field, H (A/m)

○ Current Density, J (A/m^2)

Field Value at Selected Point

x	y	z	mag
0	0	0	0

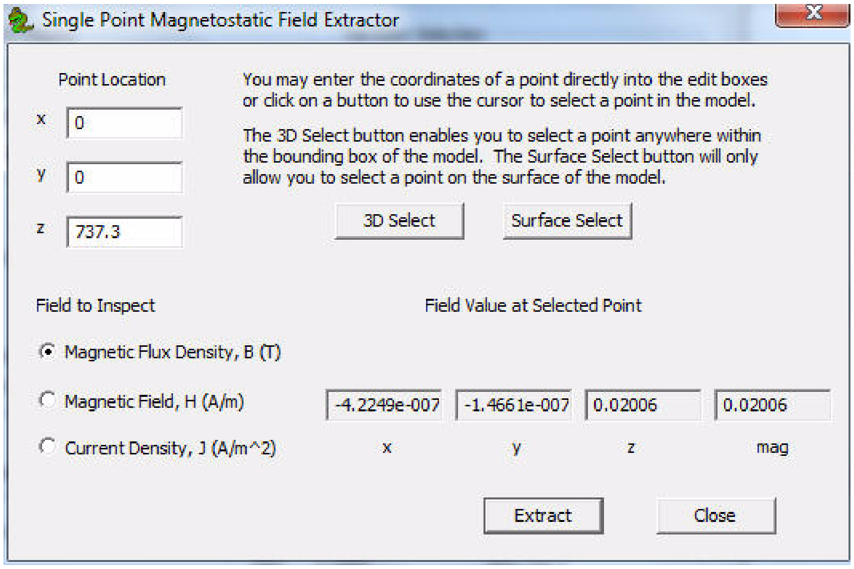
FIGURE 99. Single Point Magnetostatic Field Extractor

There are three different ways to specify the desired point:

- Direct entry - Allows you to enter the *x*, *y*, and *z* components of the point directly into the edit fields provided on the *Single Point Magnetostatic Field Extractor* dialog (*Point Location x*, *y* & *z*).
- Surface Selection - Allows you to select the point anywhere on the surface of the model. Click the *Surface Select* button and the main window containing the geometry model will become active. Use the Ctrl-Left mouse button to select the point. Only points on surfaces can be selected. A small dot will appear for the selected point and the xyz coordinates will automatically appear in the fields provided on the *Single Point Magnetostatic Field Extractor* dialog (*Point Location x*, *y* & *z*).
- 3D Selection - Allows you to select the point anywhere within the bounding box of the problem space. Click the *3D Select* button and the main window containing the geometry model will become active. A 3-plane widget will appear. Use the Ctrl-Left mouse button to move the planes to their desired positions. To complete the selection, click *3D Select* on the *Single Point Magnetostatic Field Extractor* dialog. The xyz coordinates of the point represented by the intersection of the three planes will automatically appear in the fields provided on the *Single Point Magnetostatic Field Extractor* dialog (*Point Location x*, *y* & *z*).

Once you specify the desired field and the desired point, click the *Extract* button. The resulting value(s) will appear in the appropriate *Field Value at Selected Point* boxes, as shown in Figure 100.

To exit the *Single Point Magnetostatic Field Extractor* dialog, click the *Exit* button and control returns to the *Magnetostatic Display Parameters* dialog.


Single Point Magnetostatic Field Extractor
3D Select
Surface Select
Extract
Close

Point Location

x: 0

y: 0

z: 737.3

You may enter the coordinates of a point directly into the edit boxes or click on a button to use the cursor to select a point in the model.

The 3D Select button enables you to select a point anywhere within the bounding box of the model. The Surface Select button will only allow you to select a point on the surface of the model.

Field to Inspect

● Magnetic Flux Density, B (T)

○ Magnetic Field, H (A/m)

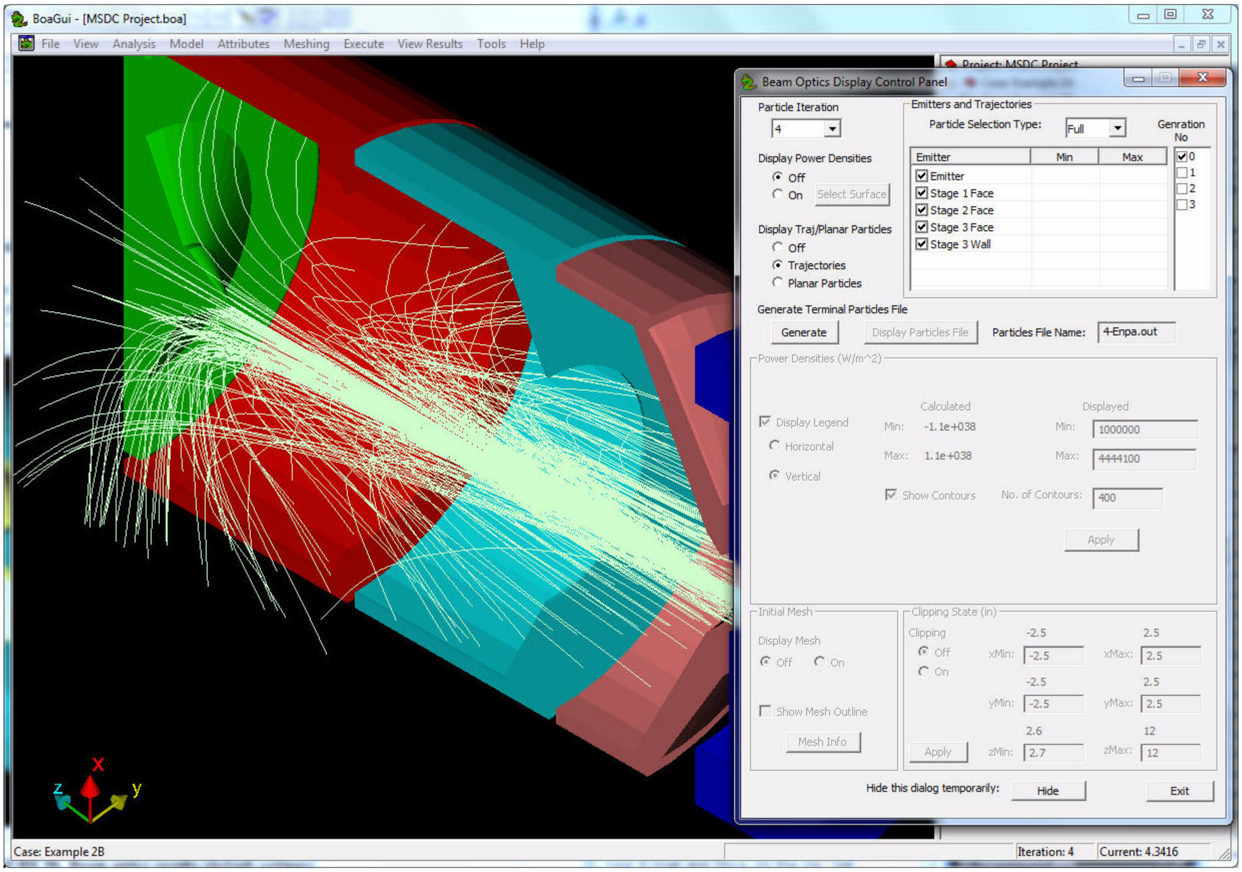
○ Current Density, J (A/m^2)

Field Value at Selected Point			
x	y	z	mag
-4.2249e-007	-1.4661e-007	0.02006	0.02006

FIGURE 100. Results from Single Point Magnetostatic Field Extraction

3.11.3 Viewing Beam Analysis Results

The **Beam Analysis** menu item will be active after completion of any analysis that includes beam optics. This menu item brings up the beam optics results in the main window. The *Beam Optics Display Control Panel* dialog automatically pops up on top, as shown Figure 101. The initially


BoaGui - [MSDC Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Project: MSDC Project
Beam Optics Display Control Panel
x
y
z
Select Surface
Generate
Display Particles File
Apply
Mesh Info
Apply
Hide
Exit

Particle Iteration: 4

Display Power Densities: ● Off ○ On

Display Traj/Planar Particles: ○ Off ● Trajectories ○ Planar Particles

Emitters and Trajectories

Particle Selection Type: Full

Emitter	Min	Max
☑ Emitter		
☑ Stage 1 Face		
☑ Stage 2 Face		
☑ Stage 3 Face		
☑ Stage 3 Wall		

Genration No: ☑0 ☐1 ☐2 ☐3

Generate Terminal Particles File

Particles File Name: 4-Enpa.out

Power Densities (W/m^2)

☑ Display Legend: ○ Horizontal ● Vertical

	Calculated	Displayed
Min:	-1.1e+038	1000000
Max:	1.1e+038	4444100

☑ Show Contours   No. of Contours: 400

Initial Mesh

Display Mesh: ● Off ○ On

☐ Show Mesh Outline

Clipping State (in)

Clipping: ● Off ○ On

xMin: -2.5 (-2.5)   xMax: 2.5 (2.5)

yMin: -2.5 (-2.5)   yMax: 2.5 (2.5)

zMin: 2.7 (2.6)   zMax: 12 (12)

Hide this dialog temporarily:

Case: Example 2B

Iteration: 4

Current: 4.3416

FIGURE 101. Beam optics results (default settings)

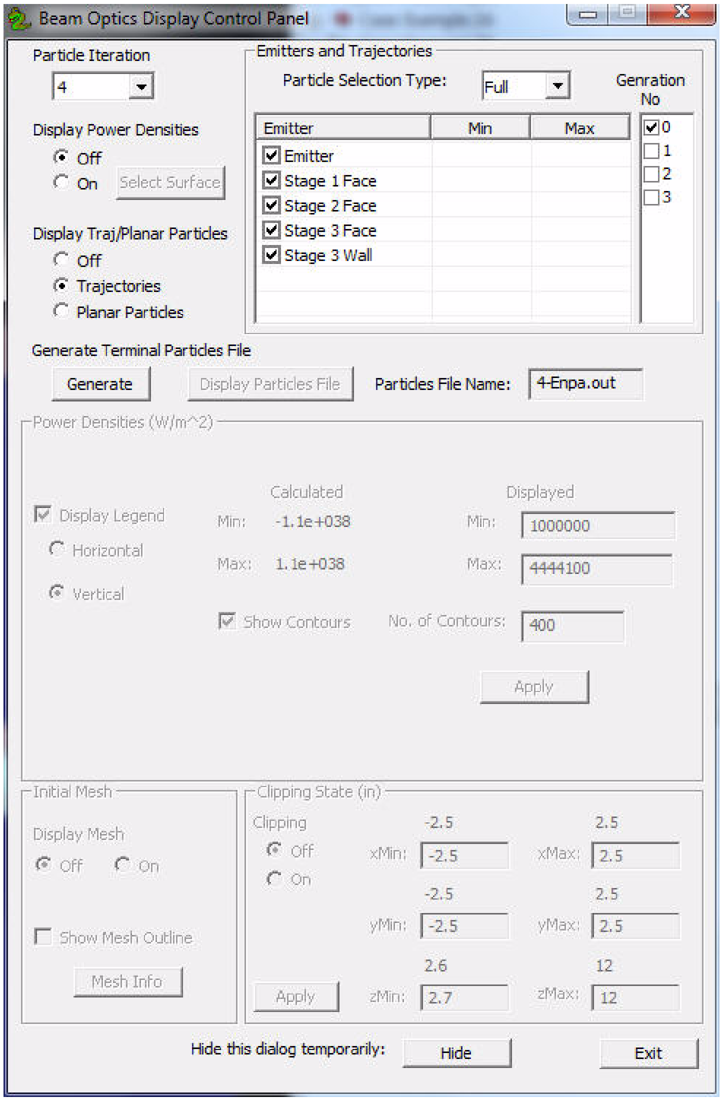
displayed results correspond to the default settings on the dialog: *Particle Iteration* set to the last saved iteration, *Display Power Densities* radio button set to ‘Off’ and *Display Traj/Planar Particles* set to ‘Trajectories’ and *Emitters and Trajectories* set to the first generation of all enabled emitters.

3.11.3.1 Overview

This *Beam Optics Display Control Panel* dialog is shown more clearly in Figure 102. The purpose of this dialog is to enable examination of the following:

- *Power densities* - This feature displays the power densities in 3D on user-selected surfaces.
- *Trajectories* - This feature displays user-selected trajectories in 3D.
- *Planar Particles* - This feature generates particle projections of user defined trajectories onto user-defined cross sections. It includes:
    - the ability to display the particle projections in 3D
    - the ability to generate more detailed information on planar particle attributes. You can save this information to a separate output file, view a 2D particle plot and view summary information.
- *Terminal Particles* - This feature generates a separate file containing the terminal attributes of all particles.

The subsequent sections describe how to use the *Beam Optics Display Control Panel* to produce results in each of these four areas. Note that this is particularly complex, feature-rich dialog and that the various sections of the dialog change depending on which radio buttons are selected.



Particle Iteration: 4

Display Power Densities: ● Off ○ On

Display Traj/Planar Particles: ○ Off ● Trajectories ○ Planar Particles

Emitters and Trajectories — Particle Selection Type: Full

Emitter	Min	Max
☑ Emitter		
☑ Stage 1 Face		
☑ Stage 2 Face		
☑ Stage 3 Face		
☑ Stage 3 Wall		

Genration No: ☑ 0 ☐ 1 ☐ 2 ☐ 3

Particles File Name: 4-Enpa.out

Power Densities (W/m^2): ☑ Display Legend ○ Horizontal ● Vertical

Calculated Min: -1.1e+038 Max: 1.1e+038

Displayed Min: 1000000 Max: 4444100

☑ Show Contours No. of Contours: 400

Initial Mesh — Display Mesh: ● Off ○ On; ☐ Show Mesh Outline

Clipping State (in) — Clipping: ● Off ○ On

xMin: -2.5 (-2.5) xMax: 2.5 (2.5)

yMin: -2.5 (-2.5) yMax: 2.5 (2.5)

zMin: 2.7 (2.6) zMax: 12 (12)

Hide this dialog temporarily:

FIGURE 102. Beam Optics Display Control Panel dialog

3.11.3.2 Iteration Selection

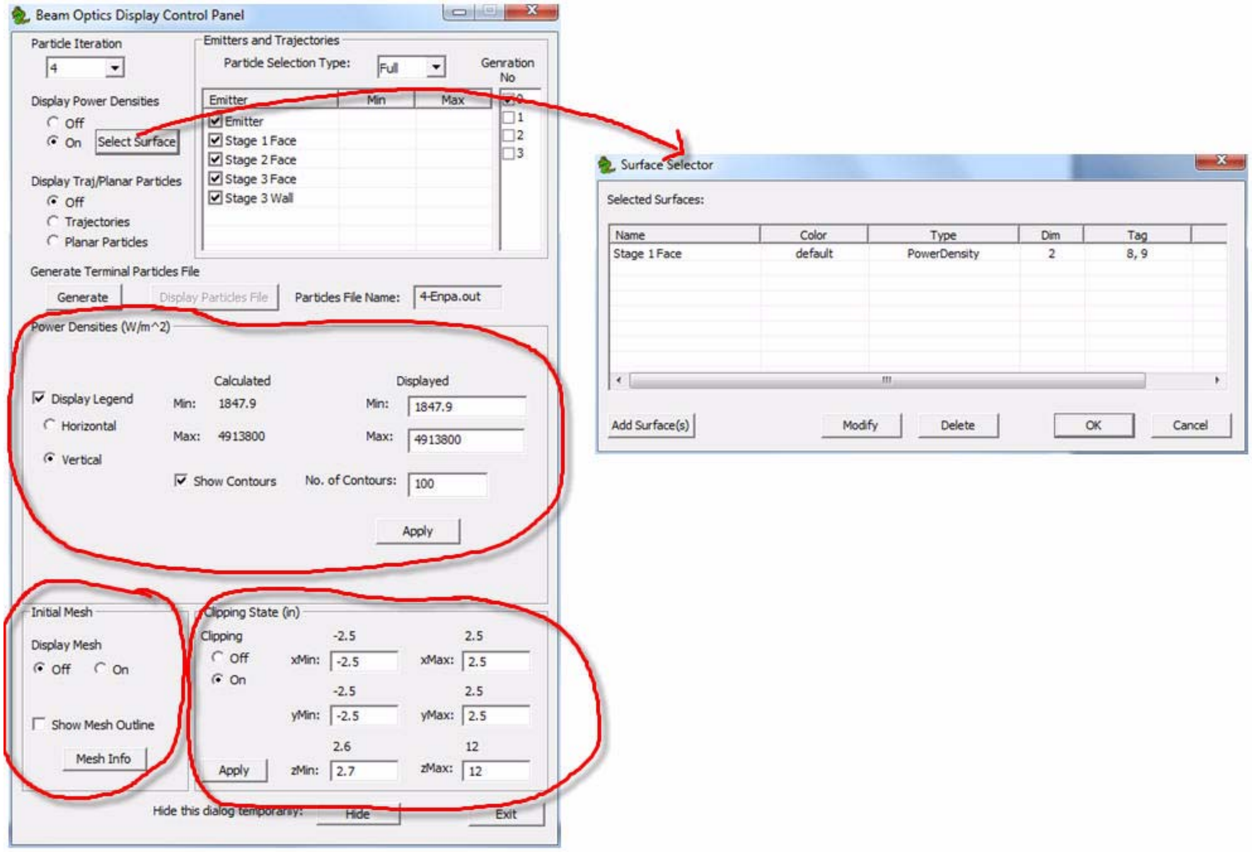
The *Particle Iteration* dropdown in the upper left corner of the *Beam Optics Display Control Panel* dialog specifies the iteration for which to display all beam optics results (i.e., Power Densities, Trajectories, Planar Particles and Terminal Particles). The contents of the dropdown box correspond to the saved iterations of the last run (as defined by the parameter *Output Solution and Source Fields Every {N} Iterations* on the *Execution Parameters* dialogs). The default iteration is the final iteration.

3.11.3.3 Power Densities

When the *Display Power Densities* radio button is set to ‘On’, a blank version of the *Surface Selector* dialog will pop up automatically if no surfaces are selected yet. This is shown below on the right side of Figure 104. In addition, the following sections become active (shown on the left side of Figure 104):

- *Power Densities*,
- *Initial (or Adapted) Mesh*, and
- *Clipping State*.

The 3D power density display will appear in the main window. Note that power densities are based on all emitters enabled in the case (i.e., all emitters enabled on the *Beam Optics* tab of the *Attribute Assignment properties* dialog). The *Emitters and Trajectories* table has no effect.


Beam Optics Display Control Panel
Select Surface
Generate
Display Particles File
Apply
Mesh Info
Apply
Hide this dialog temporarily:
Hide
Exit
Surface Selector
Add Surface(s)
Modify
Delete
OK
Cancel

Particle Iteration: 4

Display Power Densities: ○ Off ● On

Display Traj/Planar Particles: ● Off ○ Trajectories ○ Planar Particles

Emitters and Trajectories

Particle Selection Type: Full

Emitter	Min	Max
☑ Emitter		
☑ Stage 1 Face		
☑ Stage 2 Face		
☑ Stage 3 Face		
☑ Stage 3 Wall		

Genration No: ☑ 0 ☐ 1 ☐ 2 ☐ 3

Generate Terminal Particles File

Particles File Name: 4-Enpa.out

Power Densities (W/m^2)

☑ Display Legend ○ Horizontal ● Vertical

	Calculated	Displayed
Min:	1847.9	1847.9
Max:	4913800	4913800

☑ Show Contours No. of Contours: 100

Initial Mesh

Display Mesh: ● Off ○ On

☐ Show Mesh Outline

Clipping State (in)

Clipping: ○ Off ● On

xMin: -2.5 (-2.5) xMax: 2.5 (2.5)

yMin: -2.5 (-2.5) yMax: 2.5 (2.5)

zMin: 2.7 (2.6) zMax: 12 (12)

Selected Surfaces:

Name	Color	Type	Dim	Tag
Stage 1 Face	default	PowerDensity	2	8, 9

FIGURE 103. Beam Optics Display Control Panel dialog with Power Densities ‘On’

Selecting Surfaces

To add a surface on which to display power densities, click the *Select Surfaces* button next to the *Display Power Densities* ‘On’ radio button (if the dialog is not open already). Click the *Add Surface(s)* button beneath the table. This will bring up the graphics display window containing the geometry model. Rotate the model so that you can see the face of the desired surface. Select it by pressing the Ctrl-Left mouse button. The surface will change color to indicate it is selected. Press *Enter* and the surface will appear in the *Selected Surfaces* table. Add additional surfaces using the same method.

To modify the *Color* or *Name* (if editable) for a selected surface, select the table entry and click the *Modify* button. Make the desired changes in the resulting *Selected Surfaces Properties* dialog and click *OK*. To deleted a selected surface, select the desired table entry and click the *Delete* button.

To close the *Surface Selector* dialog, click *OK*.

Setting Power Density Parameters

In the *Power Densities* section, the *Show Contour* checkbox controls the appearance of the power densities on the selected surfaces.

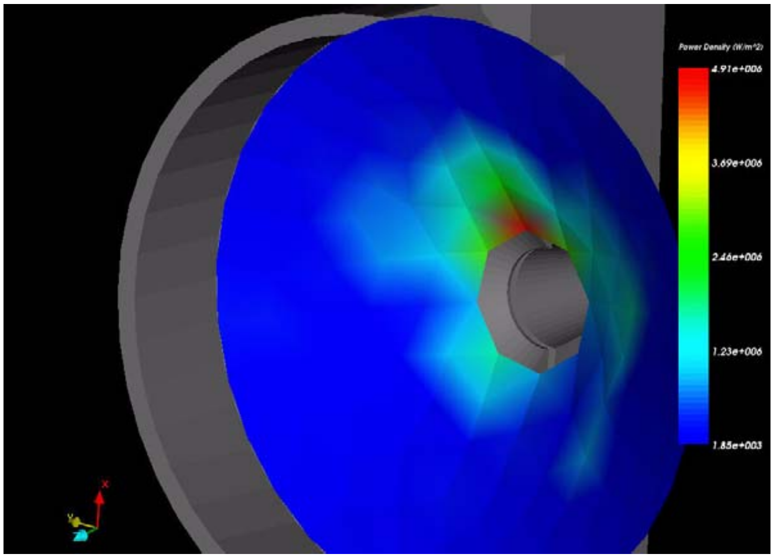


FIGURE 104. Power Density contour plot (with *Show Contours* unchecke

If the checkbox is unchecked, the power densities appears as a standard continuous contour plot like the one shown in Figure 104. Alternatively, if the *Show Contour* checkbox is checked, you specify *Number of Contours* and the display will group like values into the specified number of discrete contour lines, as shown in Figure 105.

The *Displayed Min* and *Max* values dictate the upper and lower values displayed. These values initially default to the *Calculated Min* and *Max* values. (The *Calculated Min* and *Max* values appear for reference, making it easy to reset the values.) The *Display Legend* checkbox turns the legend display on and off, and the radio buttons labeled *Horizontal* and *Vertical* dictate the legend placement.

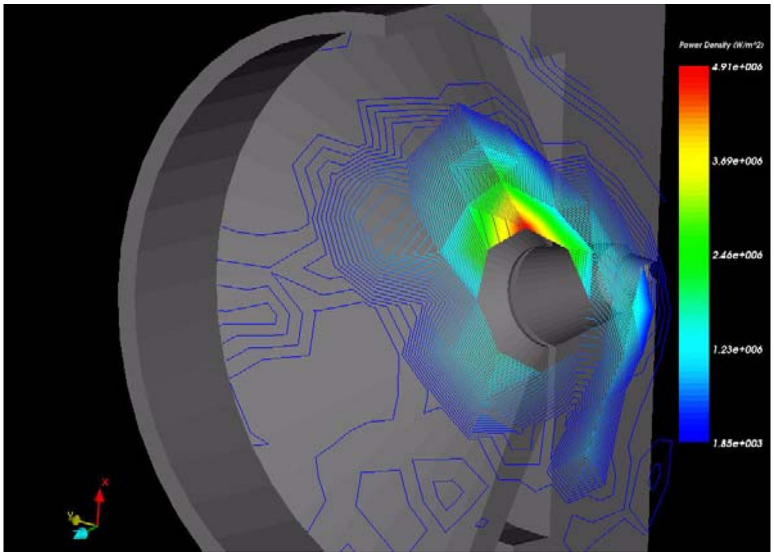


FIGURE 105. Power Density contour plot (with *Show Contours* checked

Changes take effect when you click the *Apply* button (or hit Enter).

Showing the Mesh

The *Initial Mesh (or Adapted Mesh)* section controls the display of the mesh. (The section label refers to the mesh used in the generation of the results. If the mesh was adapted, it is labeled *Adapted Mesh*; if not, it is labeled *Initial Mesh*.)

The *Display Mesh* radio buttons show and hide the mesh. The *Show Mesh Outline* checkbox displays an outline of the mesh.

Using Extent Clipping

The *Clipping State* section allows you to remove sections of the *Power Density* display, including the mesh, if it is ‘On’. It operates similarly to the Extent Clipping features discussed in the *Electrostatic* and *Magnetostatic* results sections. When the *Clipping* radio button is set to ‘On’, the XYZ ranges (*x Min, x Max, y Min*, *y Max*, *z Min*, *z Max*) are used to determine which portion of the results to display. The ranges initially default to the full range of the problem space. For reference, these default values appear above each edit box (this makes it easy to reset the values). The units for all values are the model units. Changes to the ranges do not take effect in the graphics window until you click the *Apply* button (or hit *Enter*).

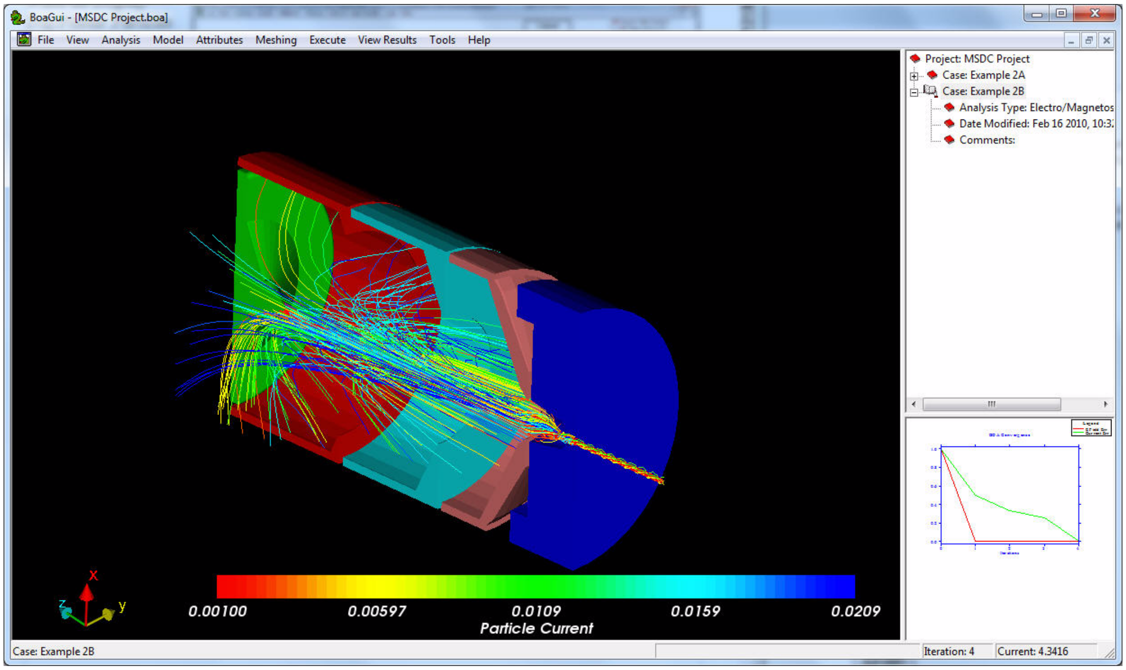
For an example of extent clipping used with the Electrostatic results, see Figure 79 (on page 68). Extent clipping for the power density display is analogous.

3.11.3.4 Trajectories

When the *Display Traj/Planar Particles* radio button is set to ‘Trajectories’:

- the trajectories appear in the main window as shown in Figure 106, and

- the *Emissions and Trajectories section* of the *Beam Optics Display Control Panel* dialog becomes active. *{confirm; only true if Mattie and Thuc take my suggestion}*


BoaGui - [MSDC Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
x
y
z
0.00100
0.00597
0.0109
0.0159
0.0209
Particle Current

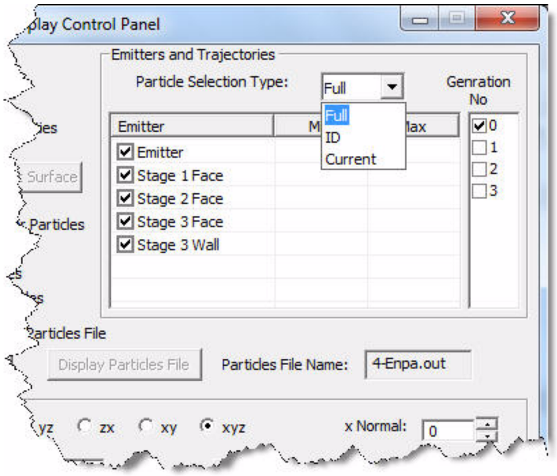
Project: MSDC Project
Case: Example 2A
Case: Example 2B
Analysis Type: Electro/Magnetos
Date Modified: Feb 16 2010, 10:3
Comments:

Case: Example 2B
Iteration: 4
Current: 4.3416

FIGURE 106. Display Trajectories ‘On’

Specifying Displayed Trajectories

The specific trajectories displayed correspond to the settings in the *Emissions and Trajectories* section, shown in Figure 107. This section includes the following controls:


play Control Panel
Emitters and Trajectories
Genration No
ies
Surface
Particles
es
es
Particles File
Display Particles File
M
lax

Particle Selection Type: Full
Full
ID
Current

Emitter
☑ Emitter
☑ Stage 1 Face
☑ Stage 2 Face
☑ Stage 3 Face
☑ Stage 3 Wall

☑ 0
☐ 1
☐ 2
☐ 3

Particles File Name: 4-Enpa.out

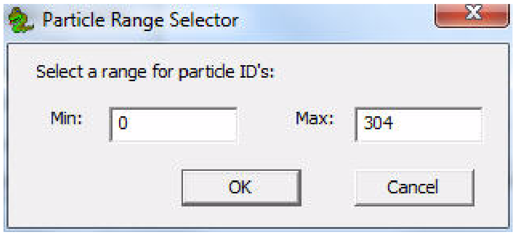
yz ○ zx ○ xy ● xyz

x Normal: 0

FIGURE 107. Emitters and Trajectories section

- *Emitters* - All enabled emitters in the case appear in the table. Emitters are turned on/off for the purposes of display via the checkbox that precedes the emitter name. At least one emitter must be checked at all times. (Emitters are defined and enabled for a case via the *Beam Optics* tab of the *Attribute Assignment Properties* dialog.)
- *Generations* - The primary generation (0th generation) as well as any secondary generations will appear in this section. Generations are turned on/off for the purposes of display via the checkbox that precedes the generation number. At least one generation must be checked at all times.
- *Particle Selection Type* - The *Particle Selection Type* dropdown box allows further refinement of which particle trajectories get included in the display:
    - ‘Full’ - This option results in the display for all particles.

- ‘ID’ - With this option, the minimum and maximum ID numbers for all particles appear in the table. To modify the ID range, click on the row for the desired emitter and the dialog shown in will appear. Enter the desired range (*Min* and *Max*) and click *OK*. Only trajectories (or planar particles) and power densities associated with the specified ID range will appear in the main window.



Select a range for particle ID's:

Min: 0

Max: 304

FIGURE 108. Particle Range Selector dialog

- ‘Current’ - With this option, the minimum and maximum current for all particles (*Min* and *Max*) appear in the table. To modify the current range, click on the row for the desired emitter and a dialog similar to will appear (however it will ask for a current range rather than an ID range). Enter the desired current range and click *OK*. (or planar particles) and power densities associated with the specified ID range will appear in the main window. *{This dialog is missing units. Tell Thuc.}*

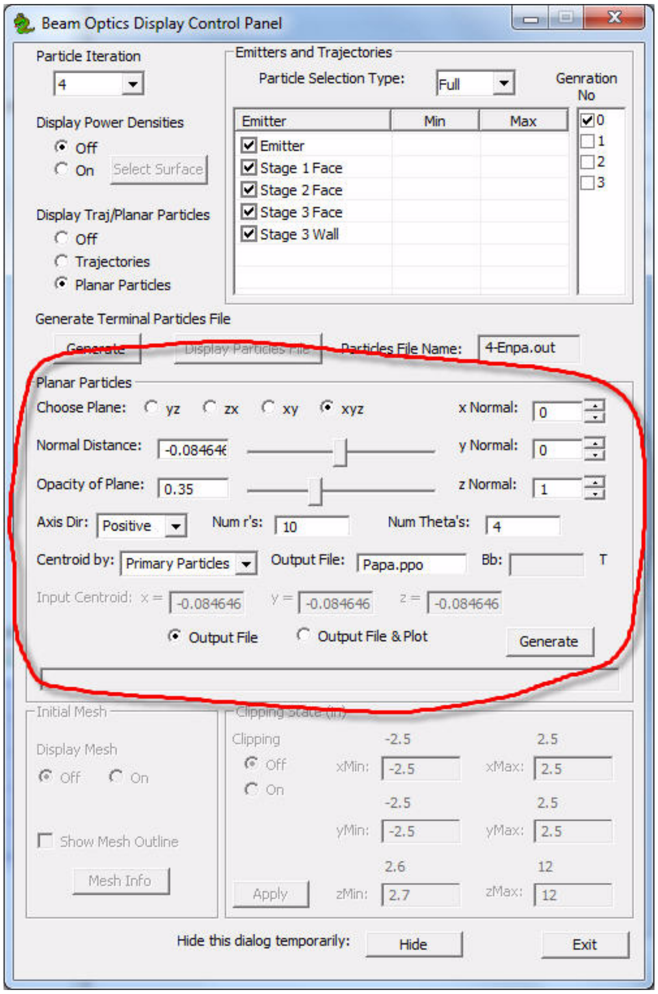
Specifying Trajectory Color

The color of the trajectories is based on the color setting in the *Colors->Trajectory* item under the **View** menu. See Figure 15 and Figure 16 (on page 17) for details on these controls. As explained there, you can change the color of the trajectories based on ID and current.

3.11.3.5 Planar Particles

When the *Display Traj/Planar Particles* radio button is set to ‘Planar Particles’:

- a cut plane appears in the main window (as shown in Figure 110)
- the *Emissions and Trajectories* section of the *Beam Optics Display Control Panel* dialog becomes active *{confirm; only true if Mattie and thuc implement my suggestion}*, and
- the *Planar Particle* section appears in the middle portion of the *Beam Optics Display Control Panel* and becomes active, as shown in red in Figure 109. This section contains input parameters needed to specify both the cut plane and to generate more detailed information about the planar particles.



Particle Iteration: 4

Display Power Densities: ● Off ○ On

Display Traj/Planar Particles: ○ Off ○ Trajectories ● Planar Particles

Particle Selection Type: Full

Emitter	Min	Max
☑ Emitter		
☑ Stage 1 Face		
☑ Stage 2 Face		
☑ Stage 3 Face		
☑ Stage 3 Wall		

Genration No: ☑ 0 ☐ 1 ☐ 2 ☐ 3

Generate Terminal Particles File — Particles File Name: 4-Enpa.out

Choose Plane: ○ yz ○ zx ○ xy ● xyz

x Normal: 0

Normal Distance: -0.08464(

y Normal: 0

Opacity of Plane: 0.35

z Normal: 1

Axis Dir: Positive

Num r's: 10

Num Theta's: 4

Centroid by: Primary Particles

Output File: Papa.ppo

Bb: T

Input Centroid: x = -0.084646 y = -0.084646 z = -0.084646

● Output File ○ Output File & Plot

Display Mesh: ● Off ○ On

☐ Show Mesh Outline

Clipping: ● Off ○ On

xMin: -2.5 (-2.5) xMax: 2.5 (2.5)

yMin: -2.5 (-2.5) yMax: 2.5 (2.5)

zMin: 2.7 (2.6) zMax: 12 (12)

Hide this dialog temporarily:

FIGURE 109. Planar Particles section of the Beam Optics Display Control Panel dialog

Selecting Emissions

The *Emissions and Trajectories* section specifies which emissions are included in the planar particle results. See the Trajectories section above (Section 3.11.3.4) for a complete explanation of this section.

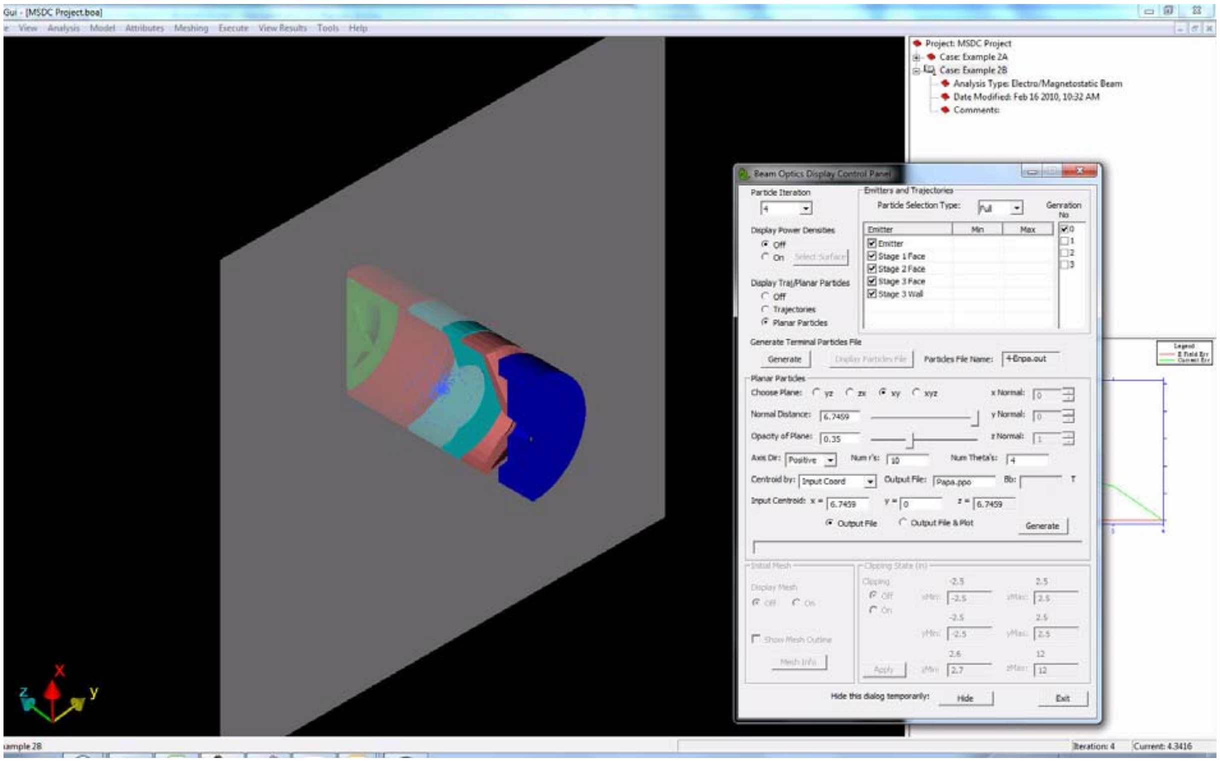
Defining the Cut Plane

In the *Planar Particles* section, the *Choose Plane* radio buttons define the orientation for the cut plane. Options include the three standard orthogonal planes and an arbitrary cut plane:

- ‘yz’ - the yz plane, with normal in the x axis,
- ‘zx’ - the zx plane, with normal in the y axis,
- ‘xy’ - the xy plane, with normal in the z axis,
- ‘xyz’ - an arbitrary plane; the three components of the vector normal to the plane are specified by the edit fields *x Normal*, *y Normal* and *z Normal*.

The *Normal Distance* slider bar defines the distance of the plane along the normal vector. The *Opacity of Plane* slider bar controls the opacity of the cut plane.

The cut plane, as defined here, appears in the main window of the BoaGui with the particles projected onto it. As you vary the above parameters, you can see the cut plane change.


Gui - [MSDC Project.boa]
View Analysis Model Attributes Meshing Execute View Results Tools Help
Project: MSDC Project
Case: Example 2A
Case: Example 2B
Beam Optics Display Control Panel
Emitters and Trajectories
Select Surface
Generate Terminal Particles File
Generate
Display Particles File
Planar Particles
Generate
Initial Mesh
Clipping State (in)
Show Mesh Outline
Mesh Info
Apply
Hide
Exit
x
y
z
ample 2B

Analysis Type: Electro/Magnetostatic Beam
Date Modified: Feb 16 2010, 10:32 AM
Comments:

Particle Iteration: 4
Display Power Densities: ● Off ○ On
Display Traj/Planar Particles: ○ Off ○ Trajectories ● Planar Particles
Particle Selection Type: Full
Genration No: ☑ 0 ☐ 1 ☐ 2 ☐ 3

Emitter	Min	Max
☑ Emitter		
☑ Stage 1 Face		
☑ Stage 2 Face		
☑ Stage 3 Face		
☑ Stage 3 Wall		

Particles File Name: 4-Enpa.out
Choose Plane: ○ yz ○ zx ● xy ○ xyz
Normal Distance: 6.7459
Opacity of Plane: 0.35
x Normal: 0
y Normal: 0
z Normal: 1
Axis Dir: Positive
Num r's: 10
Num Theta's: 4
Centroid by: Input Coord
Output File: Papa.ppo
Bb: T
Input Centroid: x = 6.7459 y = 0 z = 6.7459
● Output File ○ Output File & Plot
Display Mesh: ● Off ○ On
Clipping: ● Off ○ On
xMin: -2.5 xMax: 2.5
yMin: -2.5 yMax: 2.5
zMin: 2.7 zMax: 12
Hide this dialog temporarily:
Iteration: 4
Current: 4.3416

FIGURE 110. 3d Planar Particle Display

Generating Detailed Planar Particle Information

In addition to viewing the planar particles in the main window, you can also generate additional information, including

- a separate output file containing detailed planar particle information
- a 2D particle position plot
- summary statistics for the planar particles

To generate this additional information, first complete the necessary input parameters in the *Particle Statistics* section, including:

- *Axis Dir* - The *Axis Dir* dropdown box specifies the direction of the normal (‘positive’ or ‘negative’). When *Choose Plane* is set to the ‘xyz’ plane, then *Axis Dir* = ‘Positive’ indicates the same direction as that specified by n = (*x Normal*, *y Normal*, *z Normal*) and *Axis Dir* = ‘Negative’ indicates the opposite direction of n.
- *Num r’s* - The number of r coordinates, *[Grammar note: Thuc/Mattie, this is incorrect usage of an apostrophe; apostrophes are used to indicate possessive not plural]*
- *Num Theta’s* - The number of polar angles, *[Gammer note: Thuc/Mattie, this is incorrect usage of an apostrophe; apostrophes are used to indicate possessive not plural]*

- *Centroid* - The centroid (center of gravity) of the beam on the specified plane. The *Centroid by* dropdown box provides three options for specifying the planar beam centroid:
    - ‘All Particles’ - Uses all planar particles of all generations to compute the centroid.
    - ‘Primary Particles’ - Uses only the primary particles. (This is identical to the first method if there are no secondary emissions.)
    - ‘Input Coord’ - Allows for manual entry of the coordinates. The *Input Centroid* fields become active and you must enter two of the thee coordinates *(x,y* and *z)*. (The third coordinate is fixed, per the selected plane.)
- *Output File* - The fine name given to the output file when it is generated. (The default filename is ‘Papa.ppo’[1].) This file is stored in the case directory.

Next, select either the *Output File* or the *Output File & Plot* radio button. Then click the *Generate* button. The following will occur:

1. The output file is automatically generated and saved in the current case directory to the filename specified in the *Output File* edit box.
2. If the *Output File & Plot* radio button is selected, a plot like the one shown in Figure 111 appears in the main window. In addition, the *Plot Parameters* button (in the *Plot Statistics* section) becomes active. This button accesses the *Plot Control Parameters* dialog which enables you to modify the plot, as desired. (This dialog is shown in Figure 13, “Plot Control Panel dialog,” on page 15 and explained in more detail there.)

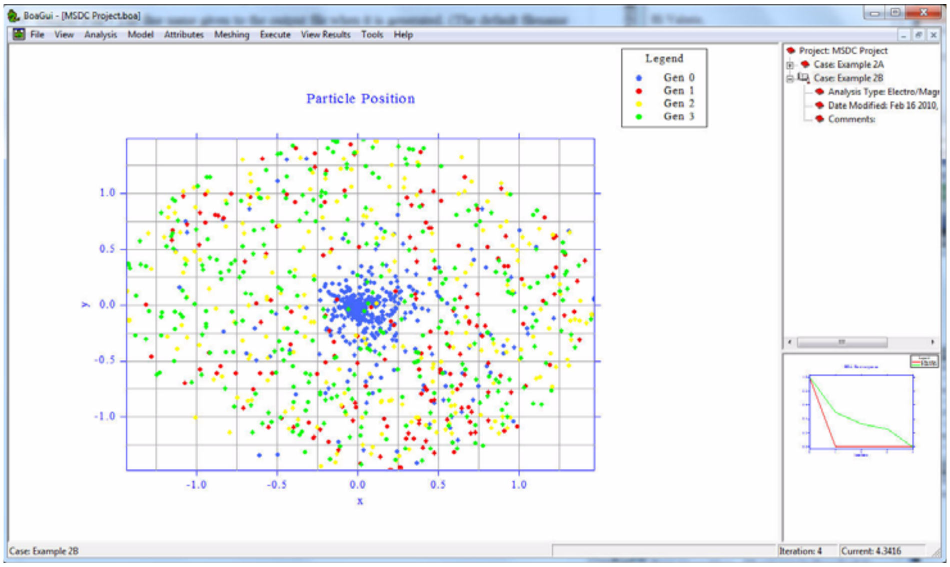

BoaGui - [MSDC Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Particle Position
Legend
● Gen 0
● Gen 1
● Gen 2
● Gen 3
1.0
0.5
y 0.0
-0.5
-1.0
-1.0
-0.5
0.0
0.5
1.0
x
Project: MSDC Project
Case: Example 2A
Case: Example 2B
Analysis Type: Electro/Magi
Date Modified: Feb 16 2010,
Comments:
Case: Example 2B
Iteration: 4
Current: 4.3416

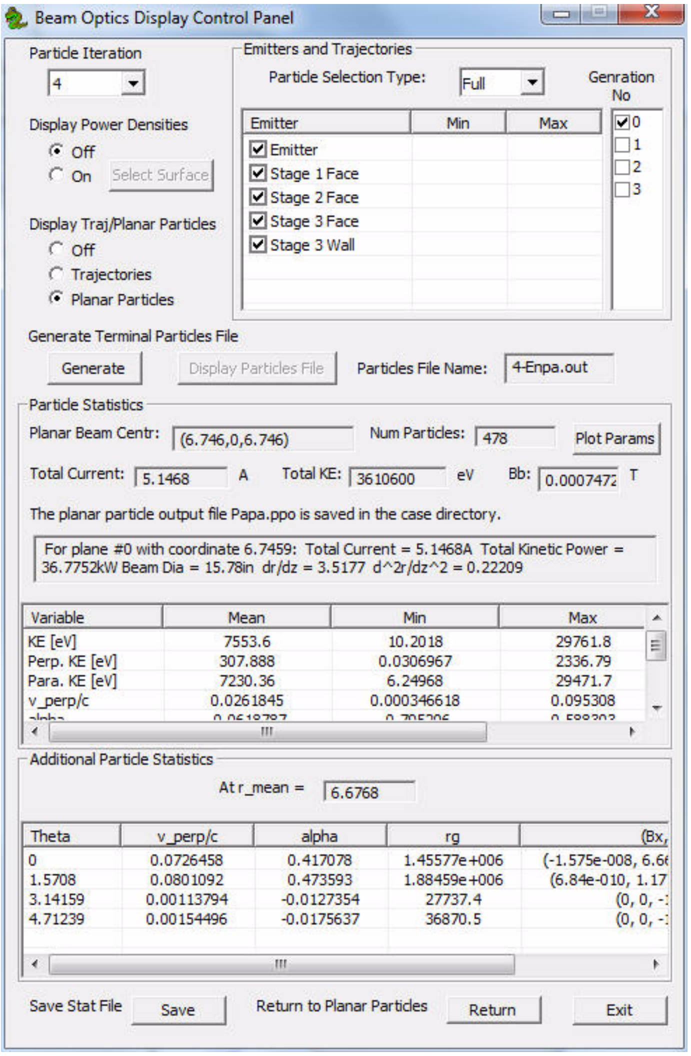
FIGURE 111. Particle Position Plot

3. The *Planar Particles* section in the middle of the *Beam Optics Display Control Panel* disappears and is replaced by the *Particle Statistics* section as shown in Figure 112. This section provides the following summary statistics:

---

[1] The term ‘Papa’ stands for Planar Particle Attributes.

- the coordinates of the centroid (*Planar Beam Centr),*
- the number of particles (*Num Particles*),
- the *Total Current*,
- the total kinetic energy (*Total KE*),
- the Brillouin field (*Bb*) - the Brillouin field produces a magnetic force that balances the space charge and centrifugal forces of the particle beam.
- In addition, a larger, non-editable box appears containing: the *Total Current, Total Kinetic Power, Beam Diameter,* the first derivative of the radius with respect to the beam axis (*dr/dz*) and the second derivative of the radius with respect to the beam axis (d^2r/dz^2). (Note that once calculated, the information in this larger, non-editable box remains available after you use the *Return* button at the bottom of the screen to switch the *Particle Statistics* section back to the *Planar Particles* section. This is also true for the Brillouin field (*Bb*).)



Particle Iteration: 4

Display Power Densities: ● Off ○ On

Display Traj/Planar Particles: ○ Off ○ Trajectories ● Planar Particles

Emitters and Trajectories

Particle Selection Type: Full

Emitter	Min	Max
☑ Emitter		
☑ Stage 1 Face		
☑ Stage 2 Face		
☑ Stage 3 Face		
☑ Stage 3 Wall		

Genration No: ☑ 0 ☐ 1 ☐ 2 ☐ 3

Generate Terminal Particles File

Particles File Name: 4-Enpa.out

Particle Statistics

Planar Beam Centr: (6.746,0,6.746)

Num Particles: 478

Total Current: 5.1468 A

Total KE: 3610600 eV

Bb: 0.0007472 T

The planar particle output file Papa.ppo is saved in the case directory.

For plane #0 with coordinate 6.7459: Total Current = 5.1468A Total Kinetic Power = 36.7752kW Beam Dia = 15.78in dr/dz = 3.5177 d^2r/dz^2 = 0.22209

Variable	Mean	Min	Max
KE [eV]	7553.6	10.2018	29761.8
Perp. KE [eV]	307.888	0.0306967	2336.79
Para. KE [eV]	7230.36	6.24968	29471.7
v_perp/c	0.0261845	0.000346618	0.095308

Additional Particle Statistics

At r_mean = 6.6768

Theta	v_perp/c	alpha	rg	(Bx,
0	0.0726458	0.417078	1.45577e+006	(-1.575e-008, 6.6
1.5708	0.0801092	0.473593	1.88459e+006	(6.84e-010, 1.17
3.14159	0.00113794	-0.0127354	27737.4	(0, 0, -
4.71239	0.00154496	-0.0175637	36870.5	(0, 0, -

Save Stat File

Return to Planar Particles

FIGURE 112. Particle Statistics section of the Beam Optics Display Control Panel dialog

If the analysis includes an external magnetic field (i.e., if the *Analysis* type is set to either ‘Electro/Magnetostatic Beam’ or ‘Electro/Magneto/Self Field Beam’), then two additional sets of statistics will appear (as shown in Figure 112):

- The lower portion of the *Particle Statistics* section will contain a table that summarizes particle statistics for the whole plane. The table includes the *Mean*, minimum (*Min*), maximum (*Max*) and standard deviation (*Sigma*) for the following:
  - *KE [eV]* - kinetic energy
  - *Perp. KE [eV]* - kinetic energy perpendicular to the beam axis
  - *Para. KE [eV]* - kinetic energy parallel to the beam axis
  - *v_perp/c* - perpendicular velocity normalized by the speed of light
  - *alpha* - the ratio of perpendicular velocity/parallel velocity
  - *B [T]* - magnetic field
  - *r [model units, e.g.,‘in’])* - beam radius

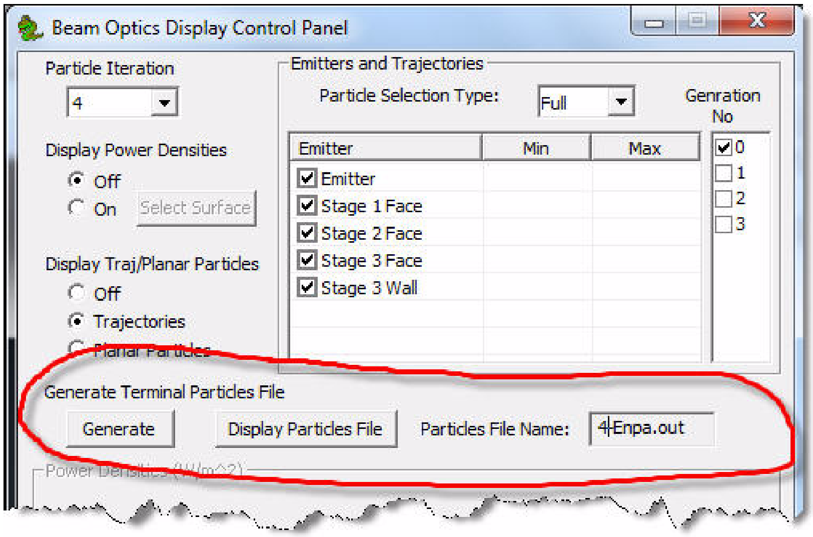
- The bottom portion of the *Beam Optics Display Control Panel* will change to a section named *Additional Particle Statistics*. This section summarizes particle statistics at the mean radius of the beam (*r Mean*). For each polar angle (*Theta*) in radians, the table provides:
  - *v_perp/c* - the perpendicular velocity normalized by the speed of light
  - *alpha* - the ratio of perpendicular velocity/parallel velocity
  - rg - the Lamor radius in model units (e.g., ‘in’). Also known as ‘gyrotron radius’.
  - *Bx*, *By*, *Bz* - the coordinates of the magnetic field, in model units (e.g., ‘in’)

  (Remember, you define the number of number of polar angles (*Num Theta’s*) in the *Planar Particle* section, as described above.)

Both sets of statistics can be saved via the *Save Stat File Save* button at the bottom of the screen.

3.11.3.6 Generate Terminal Particles

This section of the *Beam Optics Display Control Panel* dialog is used to save information on the terminal particles to a separate file. The file is generated based on the user-specified *Particle Iteration* (see dropdown box at the top of the dialog). Note that the *Emissions and Trajectories* section has no effect on the terminal particle results. All emitters enabled on the *Beam Optics* tab of the *Attribute Assignment properties* dialog are included.


Beam Optics Display Control Panel
Particle Iteration
Display Power Densities
Select Surface
Display Traj/Planar Particles
Emitters and Trajectories
Generate Terminal Particles File
Generate
Display Particles File
Power Densities (W/m^2)

Particle Iteration: 4

Display Power Densities: ● Off ○ On

Display Traj/Planar Particles: ○ Off ● Trajectories ○ Planar Particles

Particle Selection Type: Full

Emitter	Min	Max
☑ Emitter		
☑ Stage 1 Face		
☑ Stage 2 Face		
☑ Stage 3 Face		
☑ Stage 3 Wall		

Genration No: ☑ 0 ☐ 1 ☐ 2 ☐ 3

Particles File Name: 4-Enpa.out

FIGURE 114. Terminal Particles section of Beam Optics Display Control Panel dialog

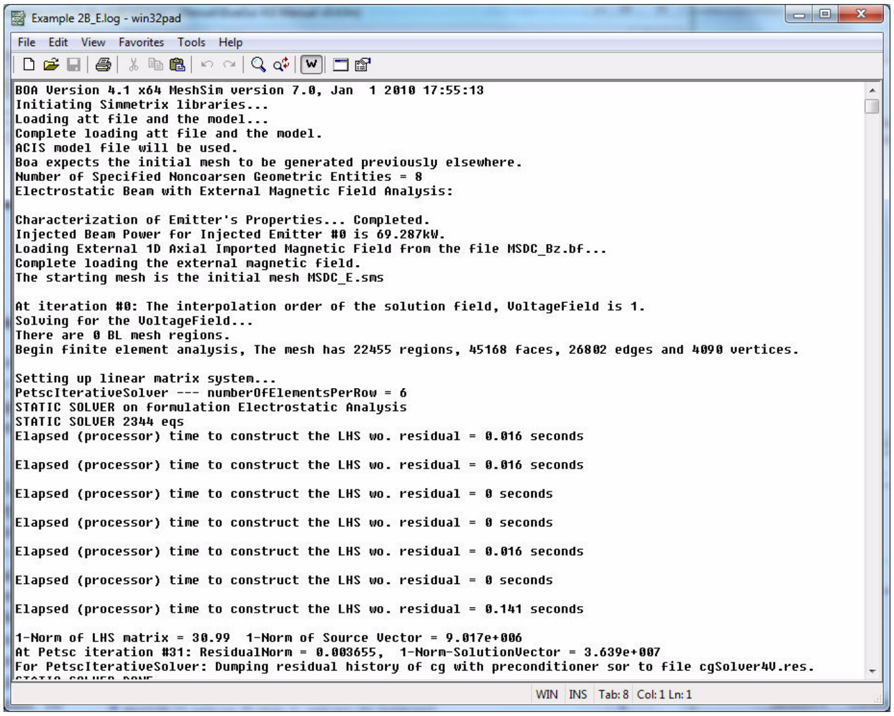
To generate the file, click the *Generate* button. An information message will pop-up showing the exact location (path and filename) for the file. The default name for the file is ‘n-Enpa.out’[1], where n corresponds to the *Particle Iteration* number. Click *OK* and the file will be created. To display the newly created file, click the *Display Particles File* button. The file will pop up in a separate window. The file format is documented in the Appendix.

---

[1] The term ‘Enpa’ stands for End Particle Attributes.

3.11.4 Viewing Run Log

This **View Run Log** menu item brings up a separate window containing the log file, as shown in Figure 115.



```
BOA Version 4.1 x64 MeshSim version 7.0, Jan  1 2010 17:55:13
Initiating Simmetrix libraries...
Loading att file and the model...
Complete loading att file and the model.
ACIS model file will be used.
Boa expects the initial mesh to be generated previously elsewhere.
Number of Specified Noncoarsen Geometric Entities = 8
Electrostatic Beam with External Magnetic Field Analysis:

Characterization of Emitter's Properties... Completed.
Injected Beam Power for Injected Emitter #0 is 69.287kW.
Loading External 1D Axial Imported Magnetic Field from the file MSDC_Bz.bf...
Complete loading the external magnetic field.
The starting mesh is the initial mesh MSDC_E.sms

At iteration #0: The interpolation order of the solution field, VoltageField is 1.
Solving for the VoltageField...
There are 0 BL mesh regions.
Begin finite element analysis, The mesh has 22455 regions, 45168 faces, 26802 edges and 4090 vertices.

Setting up linear matrix system...
PetscIterativeSolver --- numberOfElementsPerRow = 6
STATIC SOLVER on formulation Electrostatic Analysis
STATIC SOLVER 2344 eqs
Elapsed (processor) time to construct the LHS wo. residual = 0.016 seconds

Elapsed (processor) time to construct the LHS wo. residual = 0.016 seconds

Elapsed (processor) time to construct the LHS wo. residual = 0 seconds

Elapsed (processor) time to construct the LHS wo. residual = 0 seconds

Elapsed (processor) time to construct the LHS wo. residual = 0.016 seconds

Elapsed (processor) time to construct the LHS wo. residual = 0 seconds

Elapsed (processor) time to construct the LHS wo. residual = 0.141 seconds

1-Norm of LHS matrix = 30.99  1-Norm of Source Vector = 9.017e+006
At Petsc iteration #31: ResidualNorm = 0.003655,  1-Norm-SolutionVector = 3.639e+007
For PetscIterativeSolver: Dumping residual history of cg with preconditioner sor to file cgSolver4V.res.
```

FIGURE 115. Example of a Run Log

Note that the *Total Emitted Current* reported in the log file is the total current emitted by the cathode. It is <u>not</u> the actual transmitted beam current when analyzing an electron gun with grids. Electrons leave the cathode, some are turned back and some intercept the grids. To obtain the actual beam current when grids are present, use the *Display Planar Particles* feature (accessed via the *View Results*, *Beam Analysis* menu). On the *Plot Planar Particles* dialog, select a cross section plane that is normal to the beam and between the control grid and the anode. Enter the desired values in the remaining fields in the *Planar Particles* section and then click *Generate*. The *Planar Statistics* section will display the beam current.

3.11.5 Generating Case Summary

This **Generate Case Summary** menu item generates a separate file containing summary information for the case. The file is named *Case Name*_IOS.txt (where *Case Name* is the name of the current case and IOS stands for Input/Output Summary). An information message will appear showing the exact location and filename for this file.

3.11.6 Viewing Optimizer Log

{TBD: Waiting for optimizer case from M}

3.12 Tools

The **Tools** pulldown menu, shown in Figure 116, includes:

- the **Cibo** tool (a tool for creating an injected beam profile)
- the **Measure Distance** tool
- the **Material Properties** database
- the **License Installer**.

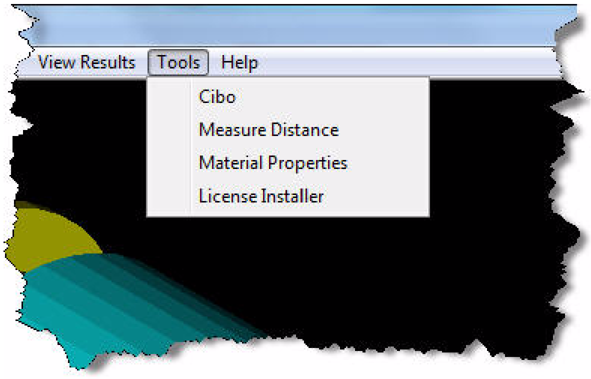
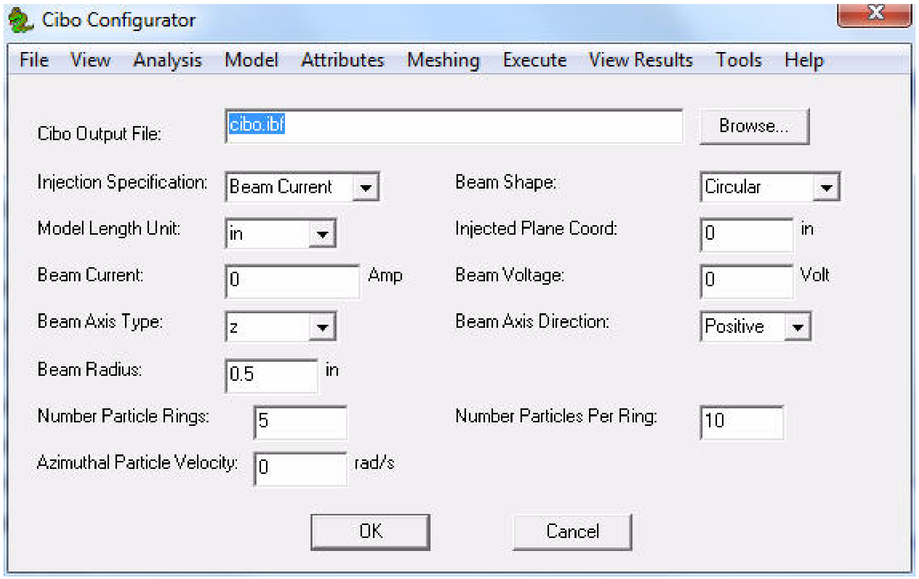

View Results
Tools
Help
Cibo
Measure Distance
Material Properties
License Installer

FIGURE 116. Tools Pull-down Menu

3.12.1 Creating an Injected Beam Profile (Cibo Tool)


Cibo Configurator
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Browse...
OK
Cancel

Cibo Output File: cibo.ibf

Injection Specification: Beam Current

Beam Shape: Circular

Model Length Unit: in

Injected Plane Coord: 0 in

Beam Current: 0 Amp

Beam Voltage: 0 Volt

Beam Axis Type: z

Beam Axis Direction: Positive

Beam Radius: 0.5 in

Number Particle Rings: 5

Number Particles Per Ring: 10

Azimuthal Particle Velocity: 0 rad/s

FIGURE 117. Cibo Configuration dialog

Cibo (Creating Injected Beam Optics) is a tool to generate an injected beam profile (a Cibo Output File) to be used by BOA for an injected emitter. It enables you to quickly and easily generate an injected beam profile in BOA format for a given set of parameters (beam voltage, current, radius, etc.) This can be useful, for example, to test a collector design without having detailed spent beam data from other RF codes.

The **Cibo** menu item brings up the *Cibo Configurator* dialog shown in Figure 117. The *Cibo Output Filename* defaults to ‘cibo.ibf’ and the default location is the current Project directory.

Note that Cibo can be used with or without a case open in BoaGui. If a case is open, BoaGui will issue a warning if you set the *Model Length Unit* to something other than the length unit defined in the case. (If no case is opened, no warning is given.) If a case is open with a specified geometry in the **Model** menu, the model length unit is the specified model's. Otherwise, the default unit is meter.

There are two options for *Injection Specification*:

- 'Beam Current' - With 'Beam Current', you specify the *Beam Current* and *Beam Voltage*. The particle charge and its energy are then calculated from the beam current, voltage and the beam geometry. Thus, each injecting particle charge varies on the injected emitter plane.
- 'Particle Charge'. With 'Particle Charge', you specify *Particle Charge* and *Particle Energy.* Each injecting particle has the same specified charge.

There are two options for *Beam Shape*:

- 'Circular' - With a 'Circular' beam shape, you specific *Beam Radius*, *Number of Particle Rings* and *Number of Particles Per Ring*. (If you specify 5 rings and 32 particles per ring, there will be a total of 160 injecting particles.)
- 'Rectangle' - With a 'Rectangular' beam shape, you specify the *Beam Height*, *Beam Width*, *Number of Vertical Particles* (along beam height) and *Number of Horizontal Particles* (along beam width). (If you specify 10 vertical and 6 horizontal, there will be 60 injecting particles.)

For the *Beam Axis Type* drop down box, select the appropriate beam axis: 'x', 'y' or 'z'. (Note that the exact label for the *Beam Height* and *Beam Width* change according to the selected *Beam Axis Type*.) You can also specify the *Azimuthal Particle Velocity* (rotation) uniformly to all injecting particles, and set the *Injected Plane Coord* (the beam axis coordinate).

3.12.2 Measuring Distance

{UPDATE THIS SECTION FOR 3D vs. SURFACE SELECT FUNCTIONALITY}

{Also, do more bug-checking: there seems to be a bug where surface select allows you to select anywhere. is this true for other surface select places in the GUI? i thought not, but check}

The *Measure Distance* feature allows you to measure the distance between two endpoints in the graphics display window. Both points must be on a part. Use the Control-Left Mouse button to select the two endpoints. Boa Gui will display the selected points as small balls. After completing selection of the second point, a message will appear showing the measured distance between the two points.

To abort the **Measure Distance** function prior to completing a measurement, hit the **Esc** key

WARNING: Before using this feature, you must set the following parameters on the **View** pull-down menu:

1. Set **View Representation** to *Orthogonal*
2. Set **Orientation** to one of the following: *Front, Back, Left, Right, Top, or Bottom*

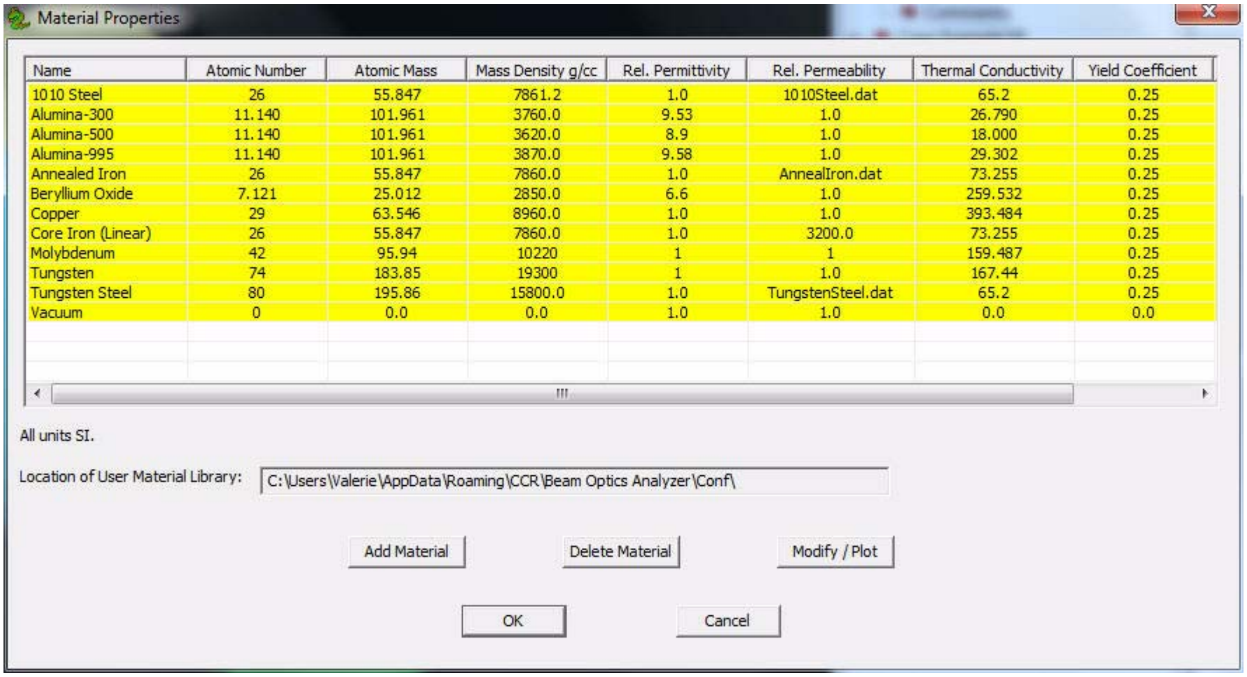
If the **View** pulldown menu parameters are not set correctly, the measured distance will be incorrect. *{TBD: Have we implemented error-checking in BoaGui? Thuc says he does not know about this, but BV explicitly told me this originally.}*

3.12.3 Material Properties

The *Materials Properties* database consists of a set of materials and the associated material properties needed for BOA analyses. These proprieties include:

- Atomic Number (used with secondary emitters)
- Atomic Mass (used with secondary emitters)
- Mass Density (used with secondary emitters)
- Relative Permittivity (used in electrostatic analyses)
- Relative Permeability (used in magnetostatic analyses)
- Thermal Conductivity (used in heat transfer analyses [not yet implemented])
- Yield Coefficient (used with secondary emitters)
- Work Function (used for thermionic emitters)

BOA provides an initial set of commonly used materials, shown in Figure 118. These initial ‘built-in’ materials cannot be edited or deleted. Users can supplement this library with their own materials. The *Location of User Material Library* shows the location where the data for all user-entered materials reside. This directory is specified once (when you initially create a project via the *New Project* dialog) and it cannot be changed thereafter.



Name	Atomic Number	Atomic Mass	Mass Density g/cc	Rel. Permittivity	Rel. Permeability	Thermal Conductivity	Yield Coefficient
1010 Steel	26	55.847	7861.2	1.0	1010Steel.dat	65.2	0.25
Alumina-300	11.140	101.961	3760.0	9.53	1.0	26.790	0.25
Alumina-500	11.140	101.961	3620.0	8.9	1.0	18.000	0.25
Alumina-995	11.140	101.961	3870.0	9.58	1.0	29.302	0.25
Annealed Iron	26	55.847	7860.0	1.0	AnnealIron.dat	73.255	0.25
Beryllium Oxide	7.121	25.012	2850.0	6.6	1.0	259.532	0.25
Copper	29	63.546	8960.0	1.0	1.0	393.484	0.25
Core Iron (Linear)	26	55.847	7860.0	1.0	3200.0	73.255	0.25
Molybdenum	42	95.94	10220	1	1	159.487	0.25
Tungsten	74	183.85	19300	1	1.0	167.44	0.25
Tungsten Steel	80	195.86	15800.0	1.0	TungstenSteel.dat	65.2	0.25
Vacuum	0	0.0	0.0	1.0	1.0	0.0	0.0

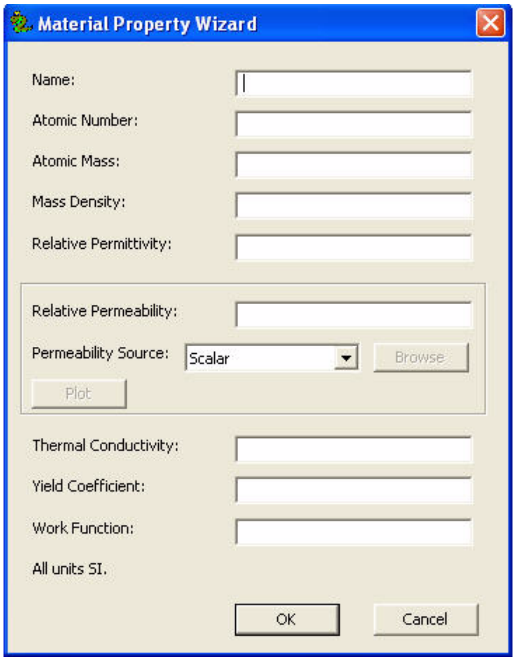
All units SI.

Location of User Material Library: C:\Users\Valerie\AppData\Roaming\CCR\Beam Optics Analyzer\Conf\

FIGURE 118. Material Properties dialog

To add a new material to the Material Properties library, click the *Add Material* button. The dialog shown in Figure 119 will appear. Enter the *Name* for the material and enter all values in SI units. Note that the *Permeability Source* dropdown box enables the *Relative Permeability* to be entered in one of three ways: ‘Scalar’, ‘File’ or ‘Formula’:

- Scalar - this option requires entry of a constant, uniform *Relative Permeability* (i.e., a single number).
- File - this option requires the filename for a two column file containing the BH curve for the material. When this option is chosen the *Browse* button becomes active so that you can locate the file. In addition, the *Plot* button becomes active so that you can plot the data. This helps ensure that the selected file contains the intended data.
- Formula *[This feature is not yet functional]* - Future versions of BOA will allow the user to enter BH curves via equations (eg B = 0.2 * H^0.2).


Material Property Wizard
Name:
Atomic Number:
Atomic Mass:
Mass Density:
Relative Permittivity:
Relative Permeability:
Browse
Plot
Thermal Conductivity:
Yield Coefficient:
Work Function:
All units SI.
OK
Cancel

Permeability Source: Scalar

FIGURE 119. Add/Edit Material dialog

Notice that some of the relative permeabilities provided for the built-in materials are scalars (i.e., constants appear in the *Relative Permeability* column in Figure 118), while others are non-linear with data provided via a separate file (i.e., filenames appear in the *Relative Permeability* column in Figure 118).

Only user entered materials can be edited or deleted. The material entries built into BOA (i.e., entries in yellow highlight) cannot be altered. To delete a user-created entry, use the *Delete Material* button (see Figure 118). Similarly to modify a user-created entry, use the *Modify Material* button.

3.12.4 License Installer

TBD {Get existing instructions}

4.0 Examples

4.1 Example 1 - JLab Klystron Gun

In this example, BOA is used to perform electrostatic and electrostatic beam analyses on an electron gun assembly used in a klystron. A cross section of the gun assembly is shown in Figure 120. The voltage is -18000 Volts on both the cathode and the focus electrode. The voltage is 0 Volts on both the anode and the mod anode. The cathode to anode spacing is 0.7 inches.

This example includes a simple electrostatic analysis without adaptive meshing (Example 1A), an electrostatic analysis with adaptive meshing (Example 1B) and an electrostatic beam analysis without adaptive meshing (Example 1C).

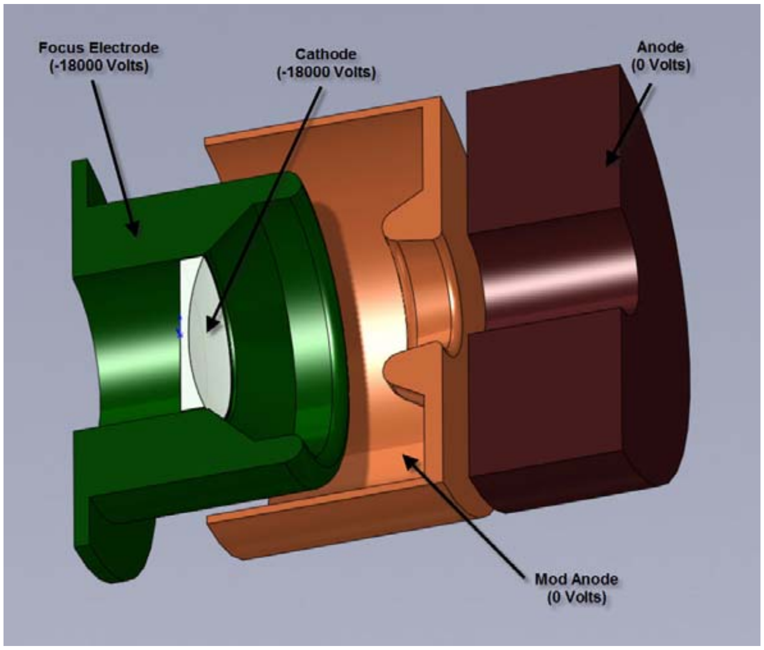


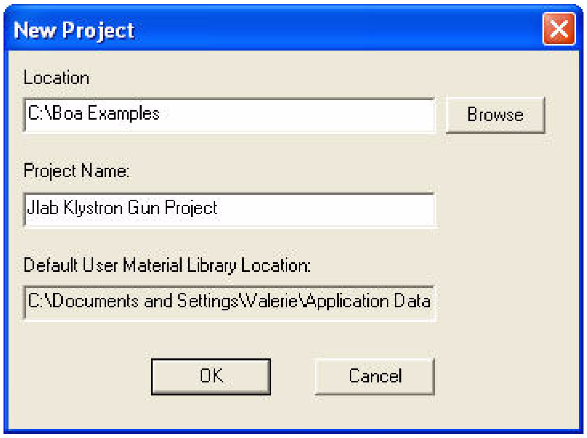
FIGURE 120. Cross section of the Electron Gun Assembly modeled in this example

4.1.1 Example 1A - Electrostatic Analysis

4.1.1.1 Getting Started

1. Open BoaGui.exe by clicking on the BoaGui icon on your desktop or the BoaGui icon in the Start menu.

4.1.1.2 Creating a New Project



Location
C:\Boa Examples

Project Name:
Jlab Klystron Gun Project

Default User Material Library Location:
C:\Documents and Settings\Valerie\Application Data

FIGURE 121. New Project Dialog Box

1. Go to the *File* pulldown menu and select *New Project*. The dialog in Figure 121 will appear. Unlike the screen shown in Figure 121, initially the two data entry boxes will be blank.
2. Use the *Browse* button to locate the desired directory for the new Project. If the desired directory does not exist yet, type in the desired name and BOAGui will created the directory for you.
3. Specify the *Project Name* in the box provided. In this example, the Project Name is 'Jlab Klystron Gun Project'.
4. Click *OK*. The new Project will appear in the Project/Case Manager tree control on the right side of the BOAGui screen. Behind the scenes, a Project folder will be created in the specified location. The project folder will contain the BOA Project file (*.boa) and all associated Case folders.

4.1.1.3 Creating a New Case

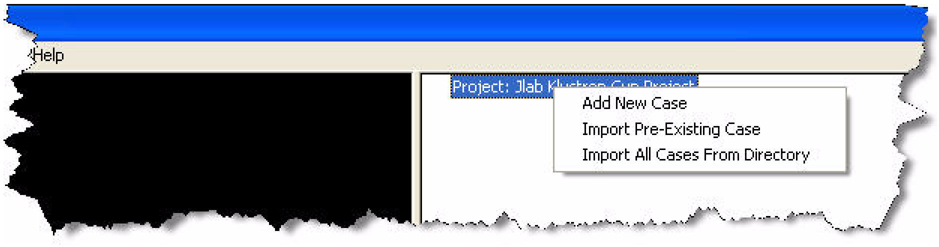
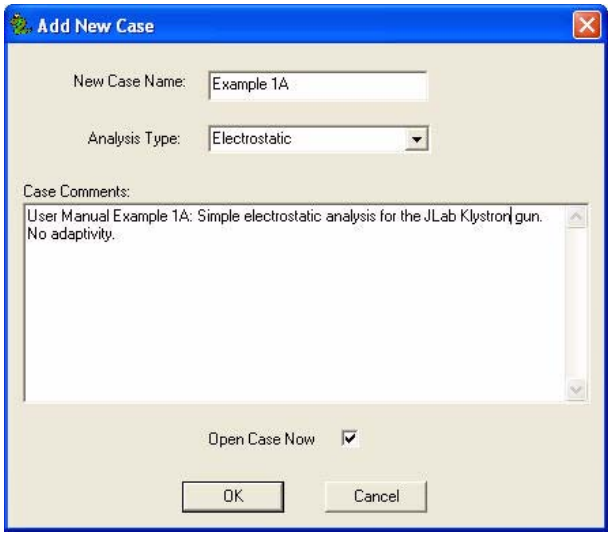

Help
Project: Jlab Klystron Gun Project
Add New Case
Import Pre-Existing Case
Import All Cases From Directory

FIGURE 122. The Project/Case Tree Control (shown with right-click menu opened)

1. Right click on 'Project Jlab Klystron Gun Project' in the Project/Case Manager tree control. The popup menu shown in Figure 122 will appear.
2. Select *Add New Case*. The dialog shown in Figure 123 will appear. Initially the data entry fields will be empty.
3. Specify the *Case Name* in the box provided. In this example the *Case Name* is 'Example 1A'.
4. Select 'Electrostatic' in the *Analysis Type* dropdown box.
5. If desired, add *Case Comments* to describe the case as shown in Figure 123.
6. Leave the *Open Case Now* checkbox checked.



New Case Name: Example 1A

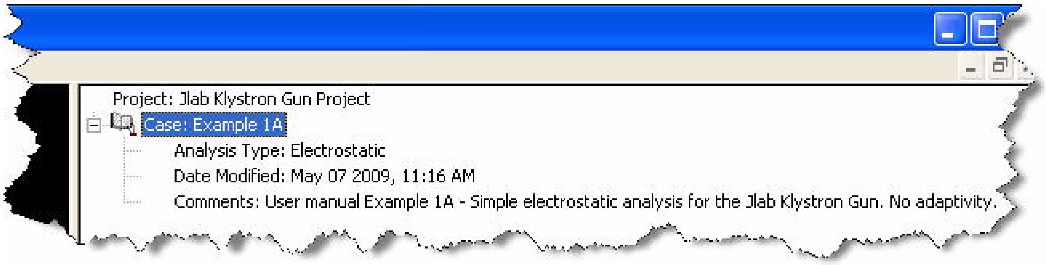
Analysis Type: Electrostatic

Case Comments:
User Manual Example 1A: Simple electrostatic analysis for the JLab Klystron gun.
No adaptivity.

Open Case Now ☑

FIGURE 123. Add New Case dialog

7. Click *OK*. The new Case will appear beneath the Project in the Project/Case Manager tree control as shown in Figure 124. Behind the scenes, a new Case folder will be placed in the current Project folder. The Case folder will include all files specific to the case.



Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
Date Modified: May 07 2009, 11:16 AM
Comments: User manual Example 1A - Simple electrostatic analysis for the Jlab Klystron Gun. No adaptivity.

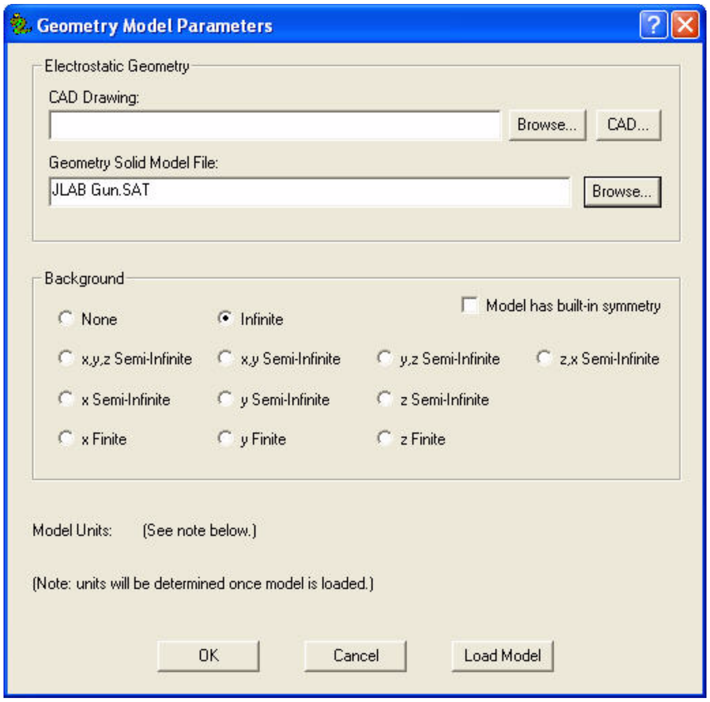
FIGURE 124. Project/Case Tree Control (with Case opened)

4.1.1.4 Saving a Case

As you continue through this example, it is a good idea to save your work occasionally. To save a case, right click on the *Case* name in the Project/Case menu bar and select *Save Case*. (Alternatively, open the *File* pulldown menu and click *Save Case*.)

4.1.1.5 Specifying the Geometry File

Next, specify the geometry file on which to base the analysis. The Boa installation includes the geometry file you will need to complete this example, as discussed below.



CAD Drawing:
Geometry Solid Model File: JLAB Gun.SAT

○ None ● Infinite ☐ Model has built-in symmetry
○ x,y,z Semi-Infinite ○ x,y Semi-Infinite ○ y,z Semi-Infinite ○ z,x Semi-Infinite
○ x Semi-Infinite ○ y Semi-Infinite ○ z Semi-Infinite
○ x Finite ○ y Finite ○ z Finite

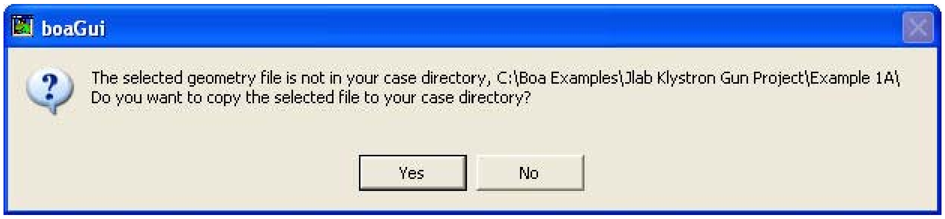
Model Units: (See note below.)

(Note: units will be determined once model is loaded.)

FIGURE 125. Geometry Model Parameters dialog

1. Click on the *Model* menu item and the dialog shown in Figure 125 will appear. (Unlike Figure 125, initially the *Geometry Solid Model File* data entry box will be blank.)
2. Click the *Browse* button next to the *Geometry Solid Model File* data entry box to locate the geometry file. This will bring up a Browse window that allows you to locate the desired geometry file.
3. Locate the file 'JLAB Gun.SAT'. If you installed BOA using the default directory settings, this file will be located in C:/Program Files/CCR/Beam Optics Analyzer/Examples/CAD Files. Select the file and click *Open* (or alternatively, double-click on the file).

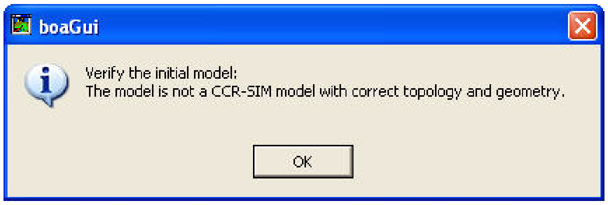
4. The informational message shown in Figure 126 will inform you that BoaGui will create a copy of the file and place it in the current Case directory. Click *Yes*.



The selected geometry file is not in your case directory, C:\Boa Examples\Jlab Klystron Gun Project\Example 1A\
Do you want to copy the selected file to your case directory?

FIGURE 126. Geometry File Copy Message

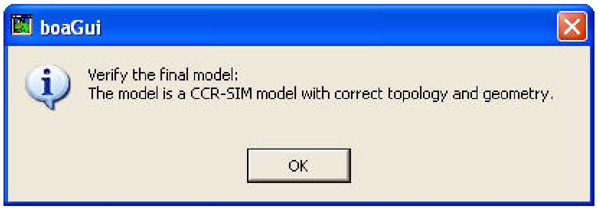
5. The name of the geometry file will appear in the *Geometry Solid Model File* entry box, as shown in Figure 125.
6. The *Model Units* will appear undefined until you load the model. Click *Load Model* and an informational message will inform you that the initial model has been loaded and verified, as shown in Figure 127. Click *OK*.



Verify the initial model:
The model is not a CCR-SIM model with correct topology and geometry.

FIGURE 127. Initial Model Verification Message

7. A second informational message will inform you that the final model has been loaded and verified as shown in Figure 128. Click *OK*. (The initial model is the model you specified, as is. The final model is the initial model plus modifications necessary to make it usable by BOA.)

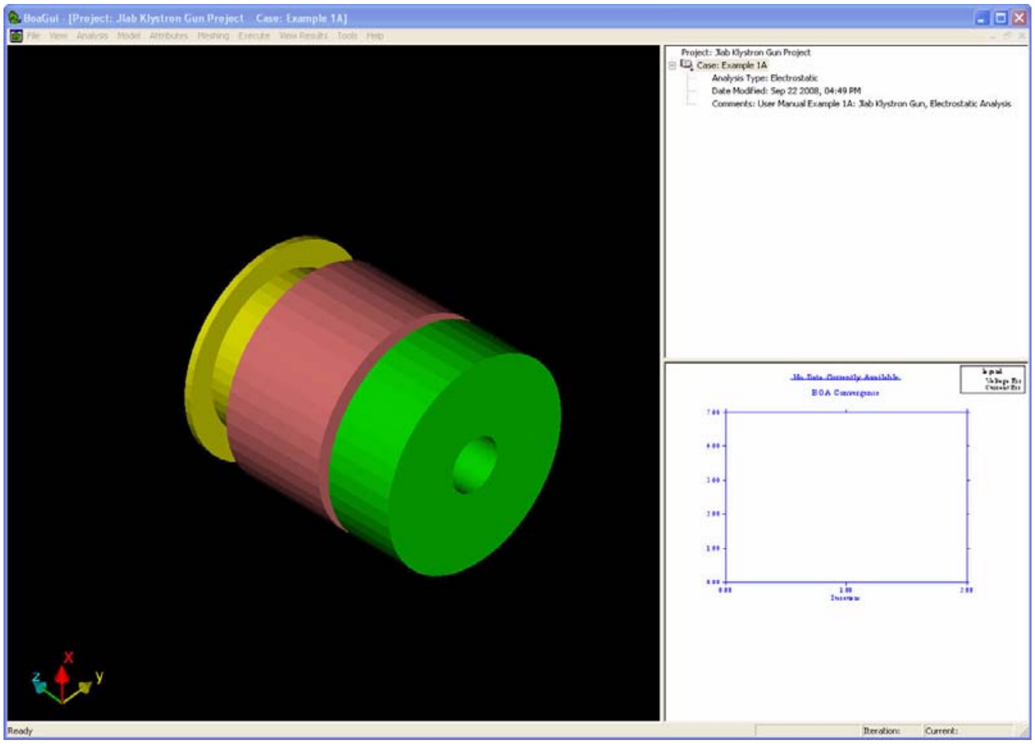


Verify the final model:
The model is a CCR-SIM model with correct topology and geometry.

FIGURE 128. Final Model Verification Message

8. The model units used in the SAT file will appear at the bottom of the *Geometry Model Parameters* dialog. In this case, the units are set to ‘inches’.
9. Click *OK* to exit the *Geometry Model Parameters* dialog.

10.The geometry model will appear in the graphics display region as shown in Figure 129.


BoaGui - [Project: Jlab Klystron Gun Project Case: Example 1A]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
X
Z
Y
No Data Currently Available
BOA Convergence
Ready
Iteration:
Current:

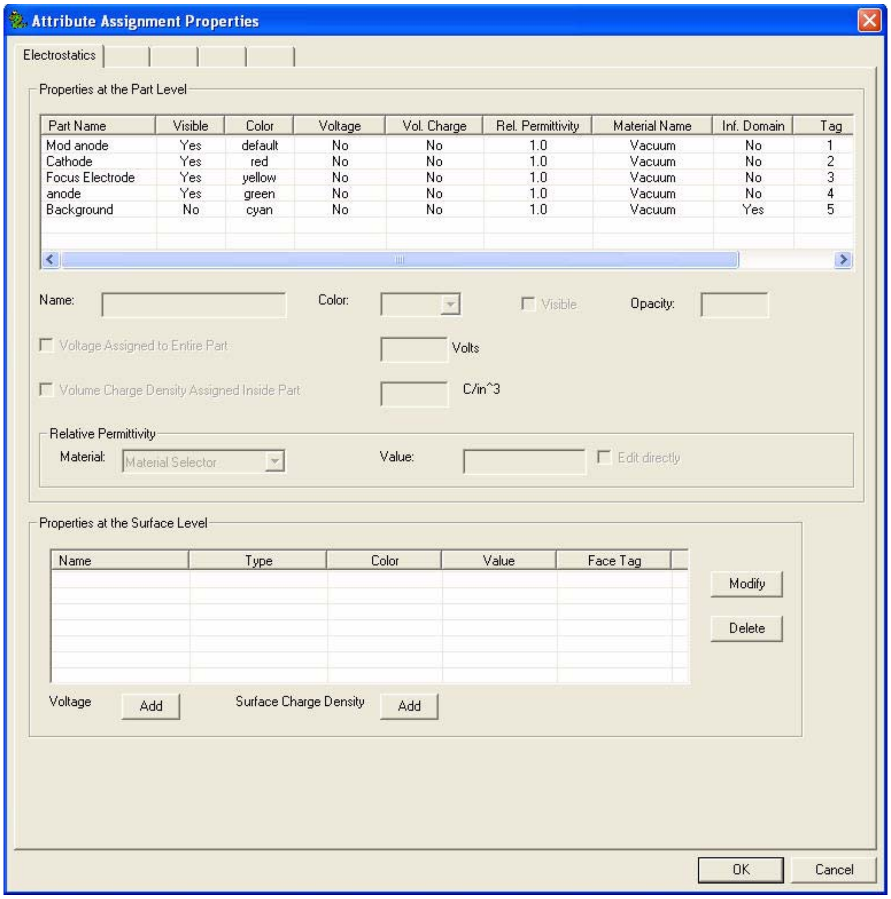
Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
Date Modified: Sep 22 2008, 04:49 PM
Comments: User Manual Example 1A: Jlab Klystron Gun, Electrostatic Analysis

FIGURE 129. Electrostatic Geometry Model (before attribute assignment)

4.1.1.6 Viewing the Geometry Model

1. Experiment with moving the model around in the graphic display region. To spin the model around, hold down the left mouse button and move the mouse around. To zoom in, hold down the right mouse button and move the mouse forward (or ‘up’). To zoom out, hold down the right mouse button and move the mouse backward (or ‘down’). You also may want to experiment with different settings on the *View* menu, such as predefined *Orientations*, and different options for *Surface Representation* (Wireframe vs. Surface) and *View Representation* (Orthogonal vs. Perspective).
2. To return to the orientation shown in Figure 129, click *View->Reset View.*

4.1.1.7 Assigning Electrostatic Attributes


Attribute Assignment Properties
Electrostatics
Properties at the Part Level
Name:
Color:
Visible
Opacity:
Voltage Assigned to Entire Part
Volts
Volume Charge Density Assigned Inside Part
C/in^3
Relative Permittivity
Material:
Material Selector
Value:
Edit directly
Properties at the Surface Level
Modify
Delete
Voltage
Add
Surface Charge Density
Add
OK
Cancel

Part Name	Visible	Color	Voltage	Vol. Charge	Rel. Permittivity	Material Name	Inf. Domain	Tag
Mod anode	Yes	default	No	No	1.0	Vacuum	No	1
Cathode	Yes	red	No	No	1.0	Vacuum	No	2
Focus Electrode	Yes	yellow	No	No	1.0	Vacuum	No	3
anode	Yes	green	No	No	1.0	Vacuum	No	4
Background	No	cyan	No	No	1.0	Vacuum	Yes	5

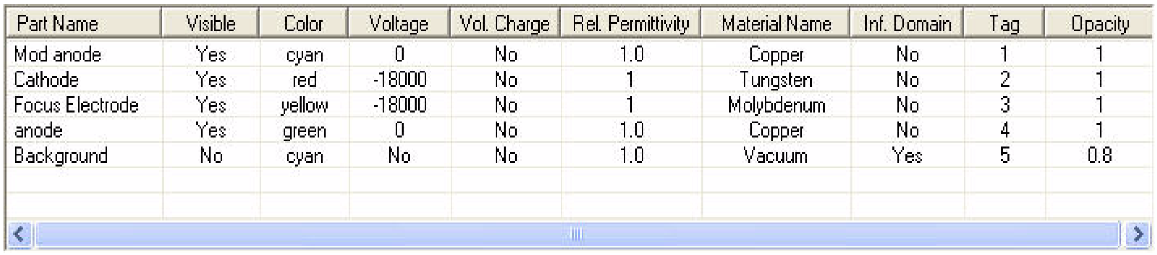
Name	Type	Color	Value	Face Tag

FIGURE 130. Attribute Assignment dialog box, Electrostatic tab

Now you are ready to begin assigning attributes.

1. Click on the *Attributes* menu item. The *Attribute Assignment Properties* dialog shown in Figure 130 will appear, with the *Electrostatics* tab active.
2. The table in the *Properties at the Part Level* section shows all parts contained in the geometry file. The table also contains the 'Background' part, if it was not already included in the geometry file. (The Background part is created by BOA based on the settings in the *Model Geometry Parameters* dialog.) The part names are identical to the names provided in the geometry file and they are not editable. Initially BoaGui will assign default attributes to all parts. Typically, the user will want to change most or all of the assigned attributes. Initially, the *Visible* checkbox is checked for all parts except the Background part.
3. Highlight the Mod Anode part. Set the *Color* dropdown box to 'cyan'. Check the *Voltage Assigned to Entire Part checkbox* and enter '0' in the data entry box. Set the *Material* dropdown box to 'Copper' in the *Relative Permittivity* section. (Note: Material assignments are not actually necessary for electrostatic or electrostatic beam analyses. Vacuum, which is the default material, produces a correct result. However, for a magnetostatic analysis, this field must be set correctly (e.g., if there is a part made of iron, the part must be assigned as such). In this example, each part is assigned a material for illustrative purposes. It is good practice to make material assignments for each part in an analysis, because then if you convert the case to include a BOA integrated magnetostatic case, the materials are already assigned.)

4. Highlight the Cathode part and assign the attributes shown in Table 1 below.

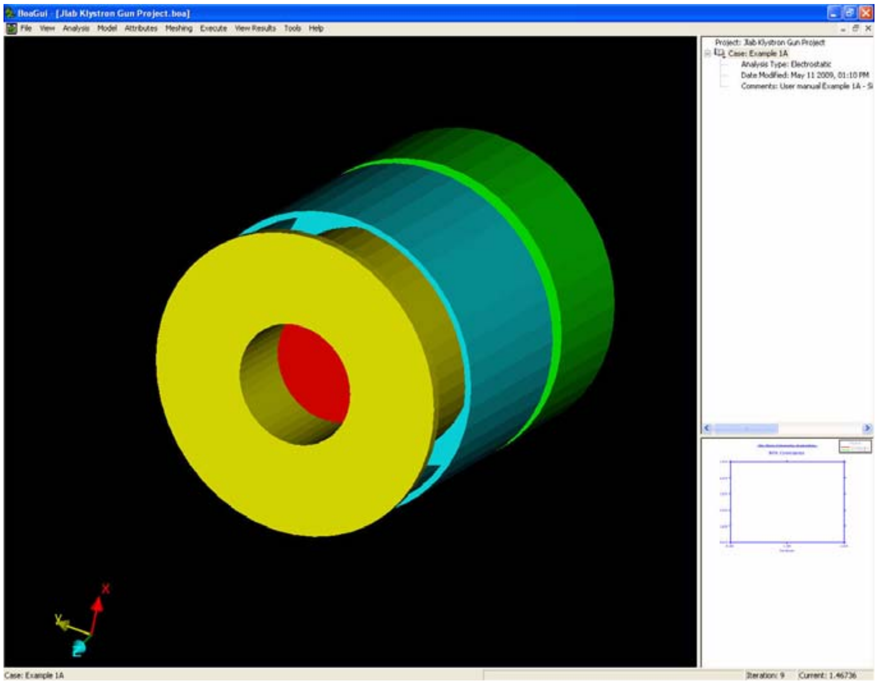


Part Name	Visible	Color	Voltage	Vol. Charge	Rel. Permittivity	Material Name	Inf. Domain	Tag	Opacity
Mod anode	Yes	cyan	0	No	1.0	Copper	No	1	1
Cathode	Yes	red	-18000	No	1	Tungsten	No	2	1
Focus Electrode	Yes	yellow	-18000	No	1	Molybdenum	No	3	1
anode	Yes	green	0	No	1.0	Copper	No	4	1
Background	No	cyan	No	No	1.0	Vacuum	Yes	5	0.8

TABLE 1. Attribute Assignments for Jlab Klystron Gun case

5. Highlight the Focus Electrode part and assign the attributes shown in Table 1 above.
6. Highlight the Anode part and assign the attributes shown in Table 1 above.
7. You have now completed the definition of properties at the part level. (The *Properties at the Surface Level* section allows you to define properties for specific surfaces on parts in the electrostatic geometry model. For this example, there will be no entries in this table.)
8. Click *OK* to close the *Attribute Assignment Properties* dialog.
9. Before proceeding, confirm that the part colors appear as shown in Figure 131. Use the mouse to orient the model so that you can see the cathode, as shown in Figure 131.
10. To return to the previous orientation, open the *View* menu and select *Reset View.*

It is a good idea to save your work occasionally. Take this opportunity to save changes to the case by right-clicking on the case name in the Project/Case menu bar and selecting *Save Case*.



Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
Date Modified: May 11 2009, 01:10 PM
Comments: User manual Example 1A - S

Case: Example 1A
Iteration: 9
Current: 1.46736

FIGURE 131. Electrostatic Model with Attributes Assigned

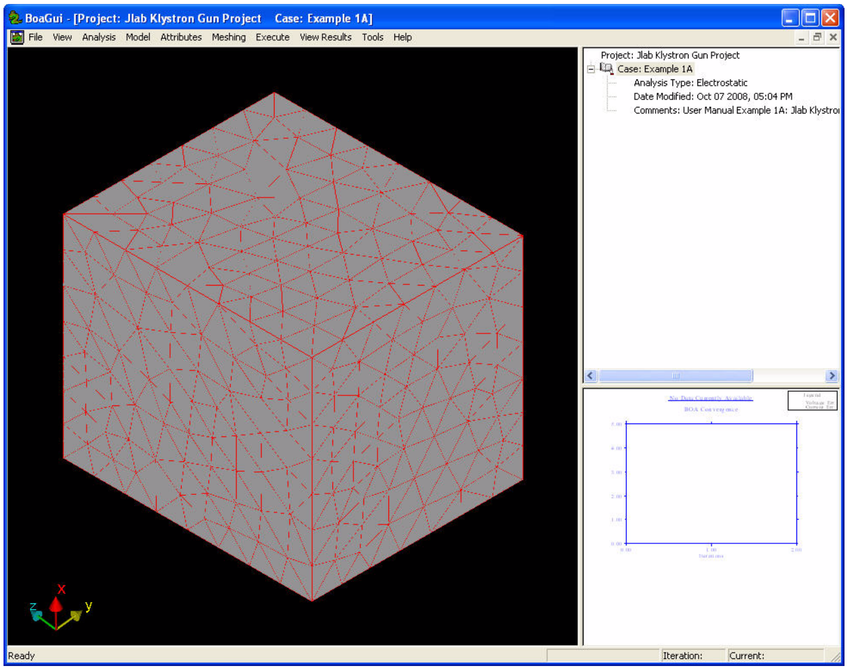
4.1.1.8 Generating the Initial Electrostatic Mesh

Once you complete the attribute assignments, you are ready to generate the initial electrostatic mesh. In this example, the default mesh parameters are sufficient so you do not need to open the *Mesh Parameters* dialog.

1. Open the *Meshing* pulldown menu and select *Generate Initial Mesh*.
2. A DOS window will appear indicating progress during the mesh generation process. When the process completes, the following text will appear at the bottom of the window: “Mesh Generation Complete, mesh file created: JLAB Gun_E.sms !Press any key to continue/exit...”. Hit any key to close the DOS window and return control back to the user interface.

4.1.1.9 Viewing the Initial Electrostatic Mesh

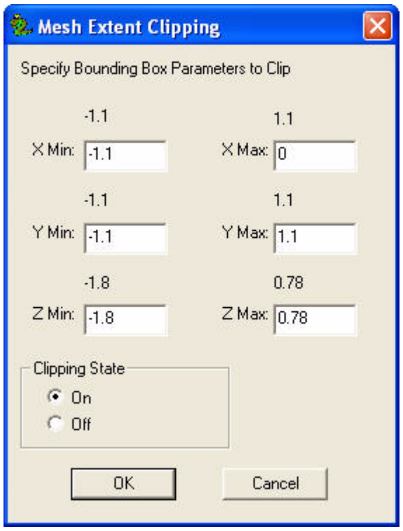
1. To view the initial mesh, open the *Meshing* pulldown menu. Select *Display Initial Mesh -> Complete*. The mesh will appear in the Main Window.
2. To get the exact orientation shown in Figure 132, click on the *View* pulldown menu and select *Reset View.* (You may need to use the zoom out feature to view the entire mesh. Right-click on the mouse and drag it backward (or down) until the complete mesh is in view.)



Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
Date Modified: Oct 07 2008, 05:04 PM
Comments: User Manual Example 1A: Jlab Klystro

FIGURE 132. Initial Electrostatic Mesh

3. To display only a part of the mesh, use the *Extent Clipping* feature. Open the *Meshing* pulldown menu and select *Extent Clipping Initial Mesh*. The *Mesh Extent Clipping* dialog will appear.
4. Change the *X Max* value to “0” as shown in Figure 133 and click *OK*. The partial mesh will appear in the graphics display region as shown in Figure 134.
5. To turn off the *Extent Clipping* feature, reopen the *Mesh Extent Clipping* dialog (i.e., open the *Meshing* pulldown menu and select *Extent Clipping Initial Mesh)* and turn the *Clipping State* radio button to ‘Off’.



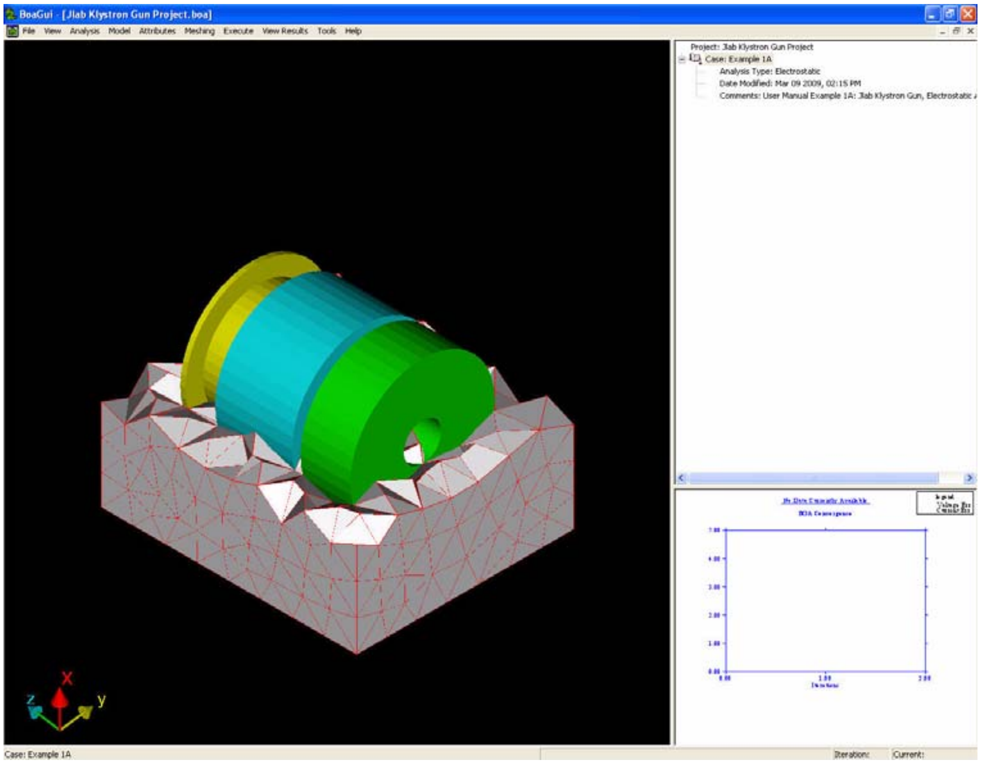
Specify Bounding Box Parameters to Clip

-1.1
X Min: -1.1
1.1
X Max: 0
-1.1
Y Min: -1.1
1.1
Y Max: 1.1
-1.8
Z Min: -1.8
0.78
Z Max: 0.78

Clipping State
● On
○ Off

FIGURE 133. Mesh Extent Clipping

6. To hide the initial mesh, open the *Meshing* pulldown menu. The *Display Initial Mesh->Complete* item is preceded by a check mark, indicating that the initial mesh is displayed. Click it to ‘uncheck’ it and the mesh display will turn off.


BoaGui - [Jlab Klystron Gun Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
x
y
z
Iteration:
Current:

Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
Date Modified: Mar 09 2009, 02:15 PM
Comments: User Manual Example 1A: Jlab Klystron Gun, Electrostatic

Case: Example 1A

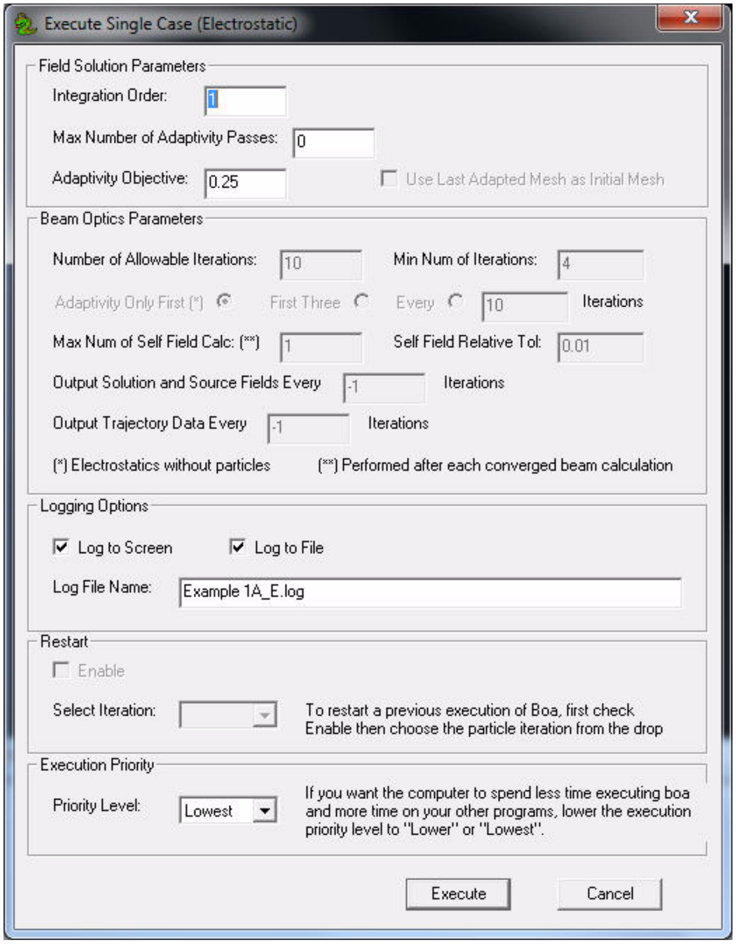
FIGURE 134. Initial Electrostatic Mesh with Extent Clipping

4.1.1.10 Executing BOA (without Adaptive Meshing)

Once the initial mesh is generated, you can execute the BOA engine.

1. Open the *Execute* pulldown menu and select *Single Case->Execute*.
2. The dialog box shown in Figure 135 will appear.

3. Leave all parameters set to the default values. (Note that *Max Number of Adaptivity Passes* is set to ‘0’which means the mesh will not be adapted for this run.) Click *Execute*.



Execute Single Case (Electrostatic)

Field Solution Parameters

Integration Order: 1

Max Number of Adaptivity Passes: 0

Adaptivity Objective: 0.25

☐ Use Last Adapted Mesh as Initial Mesh

Beam Optics Parameters

Number of Allowable Iterations: 10

Min Num of Iterations: 4

Adaptivity Only First (*) ● First Three ○ Every ○ 10 Iterations

Max Num of Self Field Calc: (**) 1

Self Field Relative Tol: 0.01

Output Solution and Source Fields Every -1 Iterations

Output Trajectory Data Every -1 Iterations

(*) Electrostatics without particles

(**) Performed after each converged beam calculation

Logging Options

☑ Log to Screen

☑ Log to File

Log File Name: Example 1A_E.log

Restart

☐ Enable

Select Iteration:

To restart a previous execution of Boa, first check Enable then choose the particle iteration from the drop

Execution Priority

Priority Level: Lowest

If you want the computer to spend less time executing boa and more time on your other programs, lower the execution priority level to "Lower" or "Lowest".

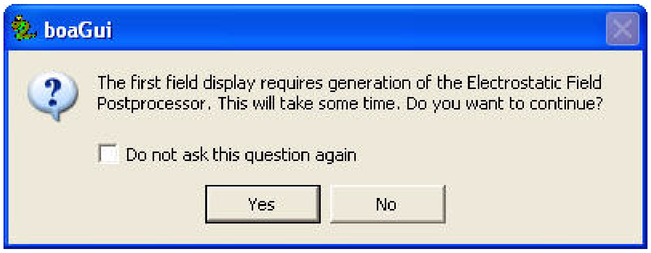
FIGURE 135. Execute Boa

4. A DOS window will appear indicating progress during BOA execution. When BOA completes, the following text will appear at the bottom of the window “!Proceed to View Results. Press any key to continue...”. Hit any key to close the DOS window and return control back to the user interface.

4.1.1.11 Viewing Electrostatic Results (without Adaptive Meshing)

After BOA completes, you must run the post processor to view the Electrostatic results.

1. Open the *View Results* pulldown menu. Select *Electrostatic*. An informational message will indicate that time is required to format the output for plotting, as shown in shown in Figure 136. Click ‘Yes’ and wait for the postprocessor to complete.



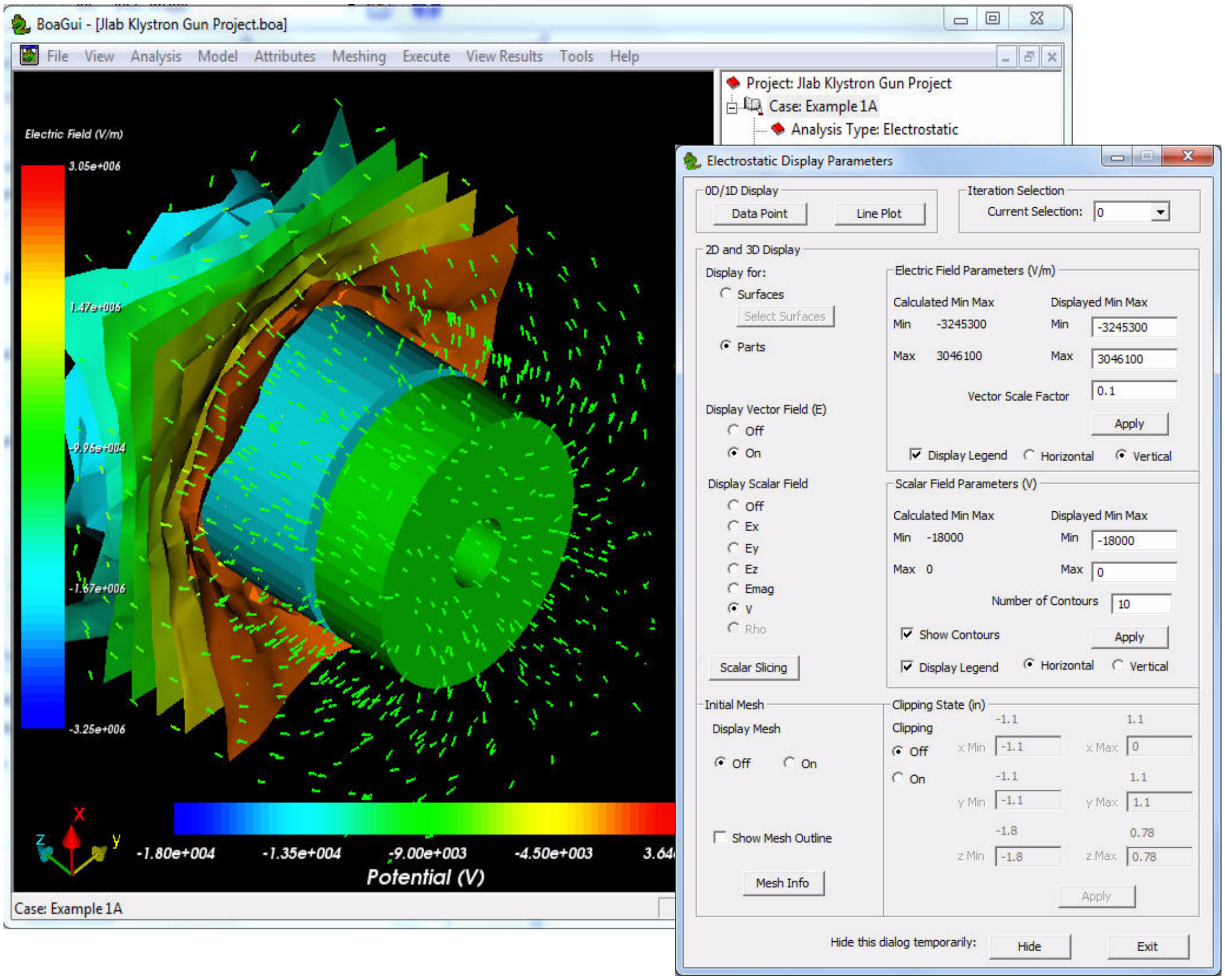
boaGui

The first field display requires generation of the Electrostatic Field Postprocessor. This will take some time. Do you want to continue?

☐ Do not ask this question again

FIGURE 136. Postprocessor Informational Message

2. The default electrostatic results will appear in the graphics display region and the *Electrostatic Display Parameters* dialog will automatically pop up on top as shown Figure 137. The dis-


BoaGui - [Jlab Klystron Gun Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Electric Field (V/m)
3.05e+006
1.47e+006
-9.96e+004
-1.67e+006
-3.25e+006
-1.80e+004
-1.35e+004
-9.00e+003
-4.50e+003
3.64
Potential (V)
x
y
z
Electrostatic Display Parameters
Data Point
Line Plot
Select Surfaces
Apply
Scalar Slicing
Mesh Info
Hide
Exit

Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic

0D/1D Display

Iteration Selection — Current Selection: 0

2D and 3D Display

Display for: ○ Surfaces ● Parts

Display Vector Field (E): ○ Off ● On

Display Scalar Field: ○ Off ○ Ex ○ Ey ○ Ez ○ Emag ● V ○ Rho

Electric Field Parameters (V/m)

Calculated Min Max		Displayed Min Max	
Min	-3245300	Min	-3245300
Max	3046100	Max	3046100

Vector Scale Factor: 0.1

☑ Display Legend ○ Horizontal ● Vertical

Scalar Field Parameters (V)

Calculated Min Max		Displayed Min Max	
Min	-18000	Min	-18000
Max	0	Max	0

Number of Contours: 10

☑ Show Contours

☑ Display Legend ● Horizontal ○ Vertical

Initial Mesh

Display Mesh: ● Off ○ On

☐ Show Mesh Outline

Clipping State (in)

Clipping: ● Off ○ On

x Min	-1.1	x Max	0
y Min	-1.1	y Max	1.1
z Min	-1.8	z Max	0.78

(Limits shown above the fields: x -1.1, 1.1; y -1.1, 1.1; z -1.8, 0.78)

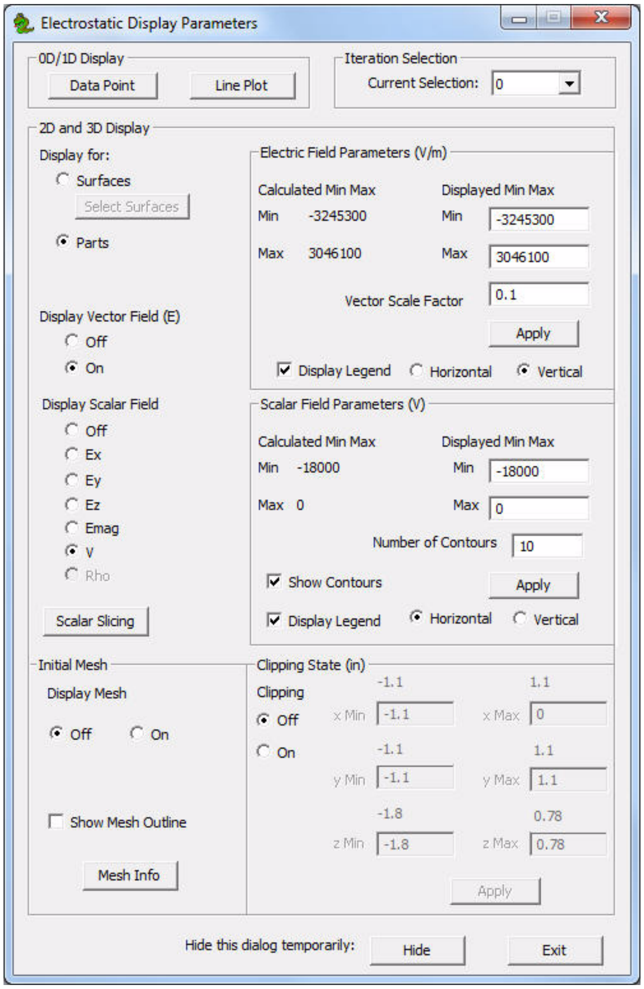
Hide this dialog temporarily:

Case: Example 1A

FIGURE 137. Electrostatic results with Voltage Contours On, Vector Field On (executed with *Max Number of Adaptivity Passes* = 0)

played results correspond to the default settings on the *Electrostatic Display Parameters* dialog: *Display Vector Field (E)* set to ‘On’ and *Display Scalar Field* set to ‘V’ (i.e., display the voltage contours). This *Electrostatic Display Parameters* dialog is shown more clearly in Figure 138.

3. Use the *Electrostatic Display Parameters* dialog to specify different outputs, as desired. Note that some sections of this dialog box require the *Apply* button for changes to take effect.


Electrostatic Display Parameters
0D/1D Display
Data Point
Line Plot
2D and 3D Display
Select Surfaces
Apply
Scalar Slicing
Mesh Info
Hide this dialog temporarily:
Hide
Exit

Iteration Selection
Current Selection: 0

Display for:
○ Surfaces
● Parts

Display Vector Field (E)
○ Off
● On

Electric Field Parameters (V/m)
Calculated Min Max: Min -3245300, Max 3046100
Displayed Min Max: Min -3245300, Max 3046100
Vector Scale Factor 0.1
☑ Display Legend ○ Horizontal ● Vertical

Display Scalar Field
○ Off
○ Ex
○ Ey
○ Ez
○ Emag
● V
○ Rho

Scalar Field Parameters (V)
Calculated Min Max: Min -18000, Max 0
Displayed Min Max: Min -18000, Max 0
Number of Contours 10
☑ Show Contours
☑ Display Legend ● Horizontal ○ Vertical

Initial Mesh
Display Mesh ● Off ○ On
☐ Show Mesh Outline

Clipping State (in)
Clipping ● Off ○ On
x Min -1.1 (-1.1), x Max 0 (1.1)
y Min -1.1 (-1.1), y Max 1.1 (1.1)
z Min -1.8 (-1.8), z Max 0.78 (0.78)

FIGURE 138. Electrostatic Display Parameters dialog

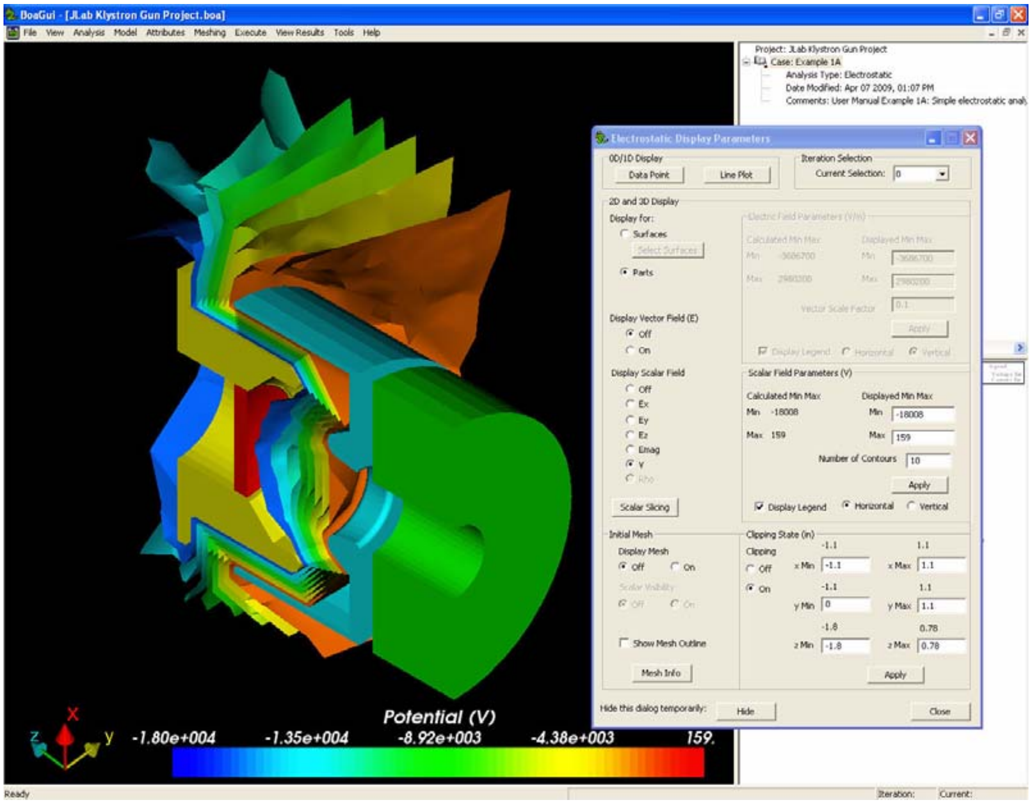
The sections below demonstrate a few examples of the different results BOA can display, including:

- 3D contour plot of the voltage field cut along zx plane (Figure 139),
- Scalar slice of the voltage field on the zx plane (Figure 141), and
- Line plot of the E magnitude along the beam axis (Figure 143)

Work through these examples to get a feel for how the how various dialogs work. Then experiment on your own to generate additional results, if desired.

4.1.1.11.1 3D contour plot of the voltage field sliced along the zx plane

Figure 139 shows a 3D contour of the voltage field sliced along the zx plane. To generate this picture, perform the following steps.


BoaGui - [JLab Klystron Gun Project.boa]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
Potential (V)
-1.80e+004
-1.35e+004
-8.92e+003
-4.38e+003
159.
x
y
z
Ready
Iteration:
Current:

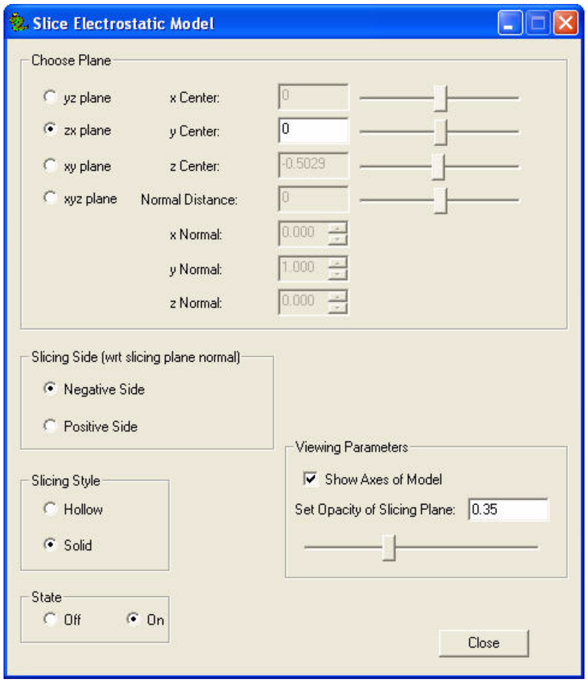
Project: JLab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
Date Modified: Apr 07 2009, 01:07 PM
Comments: User Manual Example 1A: Simple electrostatic anal

Electrostatic Display Parameters

0D/1D Display: Data Point, Line Plot
Iteration Selection: Current Selection: 0
2D and 3D Display
Display for: ○ Surfaces ● Parts
Display Vector Field (E): ● Off ○ On
Display Scalar Field: ○ Off ○ Ex ○ Ey ○ Ez ○ Emag ● V ○ Rho
Electric Field Parameters (V/m): Calculated Min Max: Min -3686700, Max 2980200; Displayed Min Max: Min -3686700, Max 2980200; Vector Scale Factor 0.1
Scalar Field Parameters (V): Calculated Min Max: Min -18008, Max 159; Displayed Min Max: Min -18008, Max 159
Number of Contours: 10
☑ Display Legend ● Horizontal ○ Vertical
Initial Mesh: Display Mesh ● Off ○ On
☐ Show Mesh Outline
Clipping State (in): Clipping ○ Off ● On
x Min -1.1, x Max 1.1
y Min 0, y Max 1.1
z Min -1.8, z Max 0.78
Hide this dialog temporarily: Hide
Close

FIGURE 139. Electrostatic results with Voltage Contours On and model and Voltage field cut along the zx plane (executed with *Max Number of Adaptivity Passes* = 0)

1. The *Electrostatic Display Parameters* dialog should still be open. (If it is not, reopen the *View Results* menu and select the *Electrostatic* item.)
2. Turn off the *Display Vector Field (E)* radio button.
3. In the *Clipping State* section, turn the *Clipping* radio button to ‘On’. Set the *x Max* value back to ‘1.1’ (you had changed it earlier in the example) and the *y Min* value to ‘0’ and then click *Apply*. This will slice the voltage field.
4. To slice the model itself, go to the *View* menu and select *Slice Model*. The dialog shown in Figure 140 will appear.
5. In the *Choose Plane* section, select the ‘zx’ radio button. Click *Close*. The main window should display the result shown in Figure 139.



Slice Electrostatic Model

Choose Plane
○ yz plane — x Center: 0
● zx plane — y Center: 0
○ xy plane — z Center: -0.5029
○ xyz plane — Normal Distance: 0
x Normal: 0.000
y Normal: 1.000
z Normal: 0.000

Slicing Side (wrt slicing plane normal)
● Negative Side
○ Positive Side

Slicing Style
○ Hollow
● Solid

State
○ Off ● On

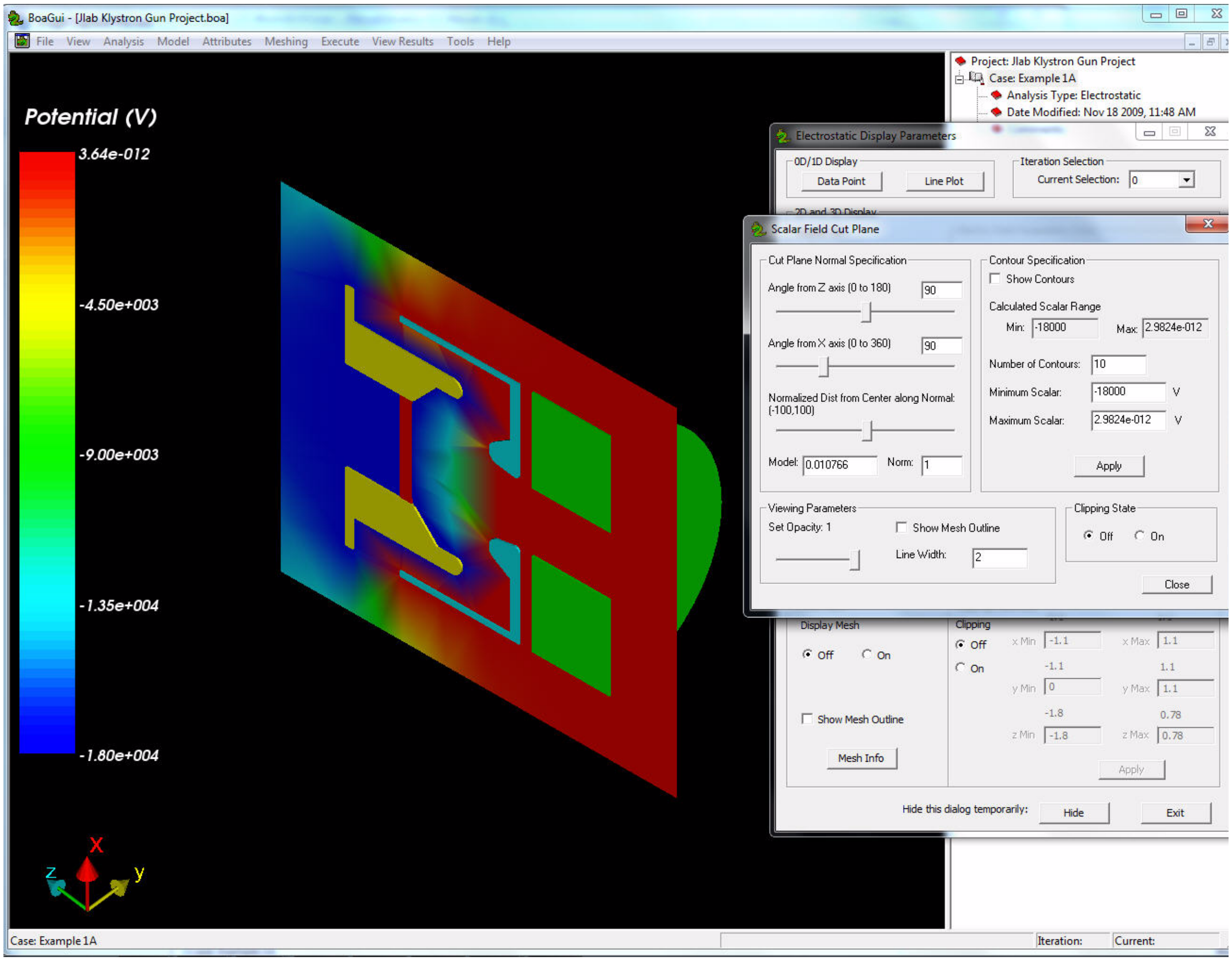
Viewing Parameters
☑ Show Axes of Model
Set Opacity of Slicing Plane: 0.35

FIGURE 140. Slice Electrostatic Model dialog

6. If desired, experiment with different parameter settings. For example, increase the *Number of Contours* to ‘20’ in the *Scalar Field Parameters* section to see how the picture changes.
7. To return the *Clipping State* to Off, set the *Clipping* radio button to ‘Off’ in the *Clipping State* section of the *Electrostatic Display Parameters* dialog.

4.1.1.11.2 Scalar slice of the voltage field on the zx plane

Figure 141 shows a scalar slice of the voltage field on the zx plane. To generate this picture, perform the following steps.


BoaGui - [Jlab Klystron Gun Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Potential (V)
3.64e-012
-4.50e+003
-9.00e+003
-1.35e+004
-1.80e+004
X
Z
y
Electrostatic Display Parameters
0D/1D Display
Data Point
Line Plot
2D and 3D Display
Scalar Field Cut Plane
Apply
Close
Mesh Info
Apply
Hide
Exit

Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
Date Modified: Nov 18 2009, 11:48 AM

Iteration Selection
Current Selection: 0

Cut Plane Normal Specification
Angle from Z axis (0 to 180) 90
Angle from X axis (0 to 360) 90
Normalized Dist from Center along Normal: (-100,100)
Model: 0.010766 Norm: 1

Contour Specification
☐ Show Contours
Calculated Scalar Range
Min: -18000 Max: 2.9824e-012
Number of Contours: 10
Minimum Scalar: -18000 V
Maximum Scalar: 2.9824e-012 V

Viewing Parameters
Set Opacity: 1
☐ Show Mesh Outline
Line Width: 2

Clipping State
● Off ○ On

Display Mesh
● Off ○ On
☐ Show Mesh Outline

Clipping
● Off
○ On
x Min -1.1 x Max 1.1
-1.1 1.1
y Min 0 y Max 1.1
-1.8 0.78
z Min -1.8 z Max 0.78

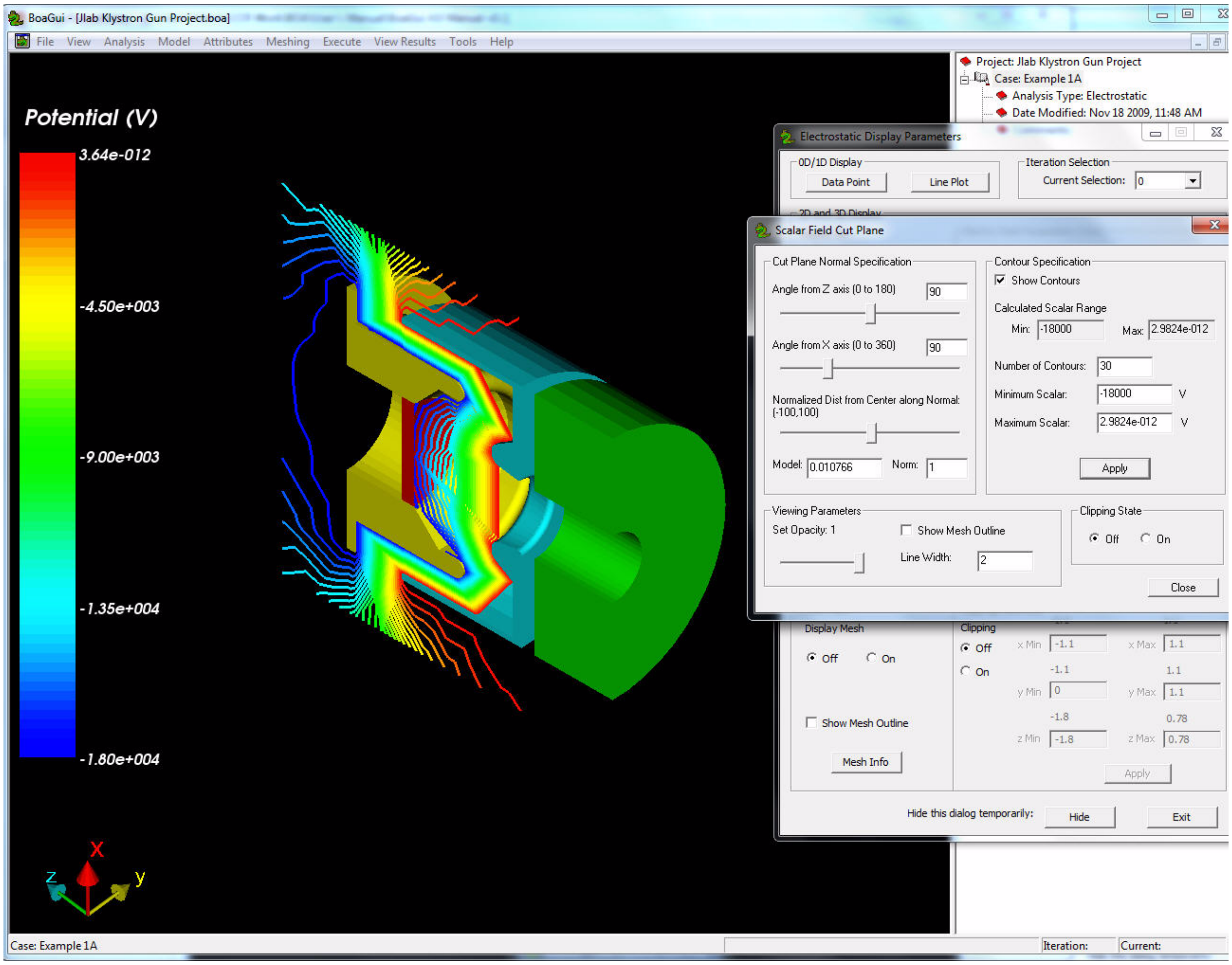
Hide this dialog temporarily:

Case: Example 1A
Iteration:
Current:

FIGURE 141. **Scalar slice of the voltage field on the zx**

1. The *Electrostatic Display Parameters* dialog should still be open.
2. Click on the *Scalar Slicing* button located beneath the *Scalar Field* radio buttons. The *Scalar Field Cut Plane* dialog will appear. (This is the top dialog shown in Figure 141.)

3. To define the cut plane as the zx plane, set the *Angle from Z axis* to ‘90’ and the *Angle from X axis* to ‘90’. Set the *Dist from Center Along Normal* slider so that *Norm* = ‘1’ (you can do this by moving the slider directly or by typing ‘1’ into the *Norm* data entry field). This allows the model to show through the cut plane slightly. The cut plane will appear as shown in Figure 141.
4. To change the cut plane to contour lines, select the *Show Contours* checkbox in the *Contour Specification* section. The plane will change to a series of contour lines.
5. Set *Number of Contours* to ‘30’and click *Apply* (or *Enter* on the keyboard). Notice the number of contour lines increase.
6. Change the *Dist from Center Along Normal* back to ‘0’ and hit *Enter* to apply the change. Notice the plane move slightly, back to center (see Figure 142).


BoaGui - [Jlab Klystron Gun Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Potential (V)
3.64e-012
-4.50e+003
-9.00e+003
-1.35e+004
-1.80e+004
X
Z
y
Electrostatic Display Parameters
0D/1D Display
Data Point
Line Plot
Iteration Selection
2D and 3D Display
Scalar Field Cut Plane
Cut Plane Normal Specification
Contour Specification
Calculated Scalar Range
Apply
Viewing Parameters
Clipping State
Close
Display Mesh
Clipping
Mesh Info
Apply
Hide
Exit
Iteration:
Current:

Project: Jlab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
Date Modified: Nov 18 2009, 11:48 AM

Current Selection: 0

Angle from Z axis (0 to 180) 90
Angle from X axis (0 to 360) 90
Normalized Dist from Center along Normal: (-100,100)
Model: 0.010766 Norm: 1

☑ Show Contours
Min: -18000 Max: 2.9824e-012
Number of Contours: 30
Minimum Scalar: -18000 V
Maximum Scalar: 2.9824e-012 V

Set Opacity: 1
☐ Show Mesh Outline
Line Width: 2

● Off ○ On

● Off ○ On
☐ Show Mesh Outline

● Off
○ On
x Min -1.1 x Max 1.1
-1.1 1.1
y Min 0 y Max 1.1
-1.8 0.78
z Min -1.8 z Max 0.78

Hide this dialog temporarily:

Case: Example 1A

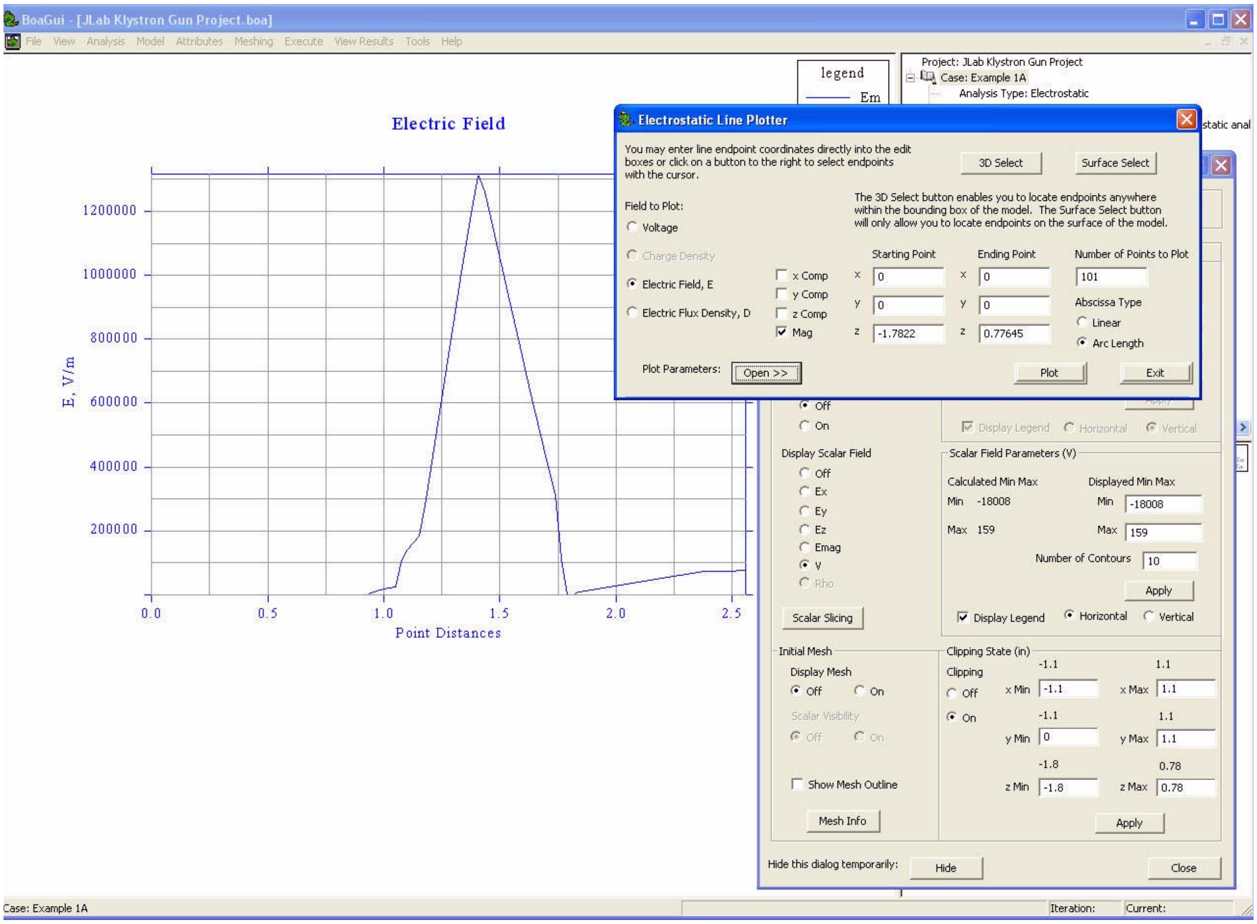
FIGURE 142. Slice Electrostatic Model dialog

7. If desired, experiment with other settings to see how the results change.
8. When you are finished experimenting, click *Close* and the cut plane will disappear.

9. To return to the complete model, click on the *View* menu and select *Slice Model*. On the resulting *Slice Electrostatic Model* dialog, select the ‘Off’ radio button in the *State* section. Click *Close* to close the dialog.

4.1.1.11.3 Line plot of the E magnitude along the beam axis (i.e., z axis)

Figure 143 shows a line plot of the E magnitude along the z (or beam) axis. To generate this picture, perform the following steps.


BoaGui - [JLab Klystron Gun Project.boa]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
Electric Field
legend
Em
E, V/m
1200000
1000000
800000
600000
400000
200000
0.0
0.5
1.0
1.5
2.0
2.5
Point Distances
Project: JLab Klystron Gun Project
Case: Example 1A
Analysis Type: Electrostatic
static anal
Electrostatic Line Plotter
3D Select
Surface Select
Plot
Exit
Scalar Slicing
Apply
Mesh Info
Hide
Close
Case: Example 1A
Iteration:
Current:

Electrostatic Line Plotter

You may enter line endpoint coordinates directly into the edit boxes or click on a button to the right to select endpoints with the cursor.

The 3D Select button enables you to locate endpoints anywhere within the bounding box of the model. The Surface Select button will only allow you to locate endpoints on the surface of the model.

Field to Plot:
○ Voltage
○ Charge Density
● Electric Field, E
○ Electric Flux Density, D

☐ x Comp
☐ y Comp
☐ z Comp
☑ Mag

	Starting Point	Ending Point
x	0	0
y	0	0
z	-1.7822	0.77645

Number of Points to Plot: 101

Abscissa Type
○ Linear
● Arc Length

Plot Parameters: Open >>

● Off
○ On

☑ Display Legend ○ Horizontal ● Vertical

Display Scalar Field
○ Off
○ Ex
○ Ey
○ Ez
○ Emag
● V
○ Rho

Scalar Field Parameters (V)

	Calculated Min Max	Displayed Min Max
Min	-18008	-18008
Max	159	159

Number of Contours: 10

☑ Display Legend ● Horizontal ○ Vertical

Initial Mesh
Display Mesh: ● Off ○ On
Scalar Visibility: ● Off ○ On
☐ Show Mesh Outline

Clipping State (in)
Clipping: ○ Off ● On

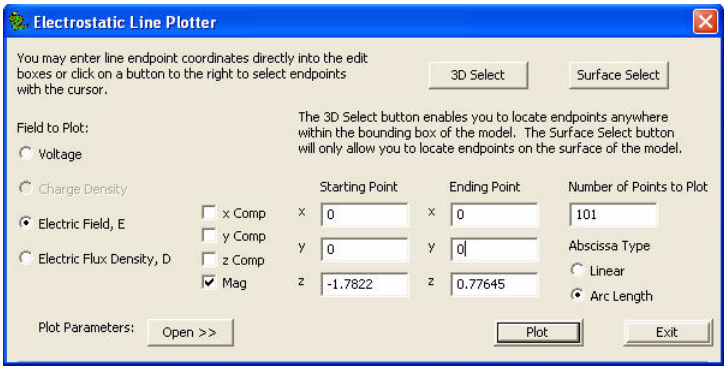
	Min		Max	
x	-1.1	-1.1	1.1	1.1
y	-1.1	0	1.1	1.1
z	-1.8	-1.8	0.78	0.78

Hide this dialog temporarily: Hide

FIGURE 143. Line plot of E magnitude along beam axis (i.e., z

1. The *Electrostatic Display Parameters* dialog should still be open.

2. Click on the *Line Plot* located in the *0D/1D Display* section at the top of the dialog. The *Electrostatic Line Plotter* dialog will appear.
3. Change the *Field to Plot* radio button to *Electric Field, E* and leave the *Mag* checkbox checked.
4. Set the *Starting Point* and *Ending point* for both *x* and *y* to ‘0’. Leave the *Starting Point* and *Ending Point* for *z* at the default.



You may enter line endpoint coordinates directly into the edit boxes or click on a button to the right to select endpoints with the cursor.

The 3D Select button enables you to locate endpoints anywhere within the bounding box of the model. The Surface Select button will only allow you to locate endpoints on the surface of the model.

Field to Plot:
○ Voltage
○ Charge Density
● Electric Field, E
○ Electric Flux Density, D

☐ x Comp
☐ y Comp
☐ z Comp
☑ Mag

	Starting Point	Ending Point
x	0	0
y	0	0
z	-1.7822	0.77645

Number of Points to Plot: 101

Abscissa Type
○ Linear
● Arc Length

Plot Parameters:

FIGURE 144. Electrostatic Line Plotter

5. Set the *Abscissa Type* radio button to ‘Arc Length’.
6. Click *Plot*. The line plot will appear in the main window, as shown in Figure 143.
7. If desired, experiment with other settings to view different results.
8. Click *Exit* to close the *Electrostatic Line Plotter* dialog.
9. Click *Exit* to close the *Electrostatic Display Parameters* dialog.
10. Before continuing on to Example 1B in the next section, make sure to save the case. (Right-click on the case name in the Project/Case Tree Control and select *Save Case*.)
11. Close the Case. (Right-click on the case name in the Project/Case Tree Control and select *Close Case*.)

4.1.2 Example 1B - Electrostatic Analysis (with Adaptive Meshing)

Next, you will rerun BOA with adaptive meshing turned on to generate a more accurate solution.

4.1.2.1 Creating a New Case

To begin, you will copy the existing case Example 1A to a new case, as follows.

1. Right click on the ‘Case: Example 1A’ case in the Project/Case Manager tree control. The sub-menu shown in Figure 145 will appear.

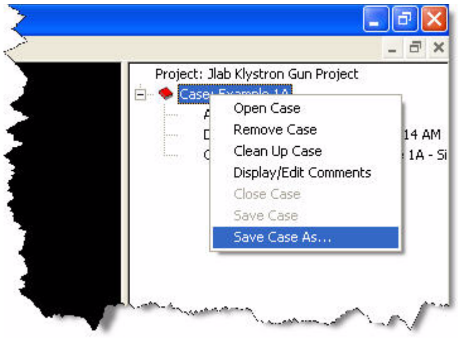
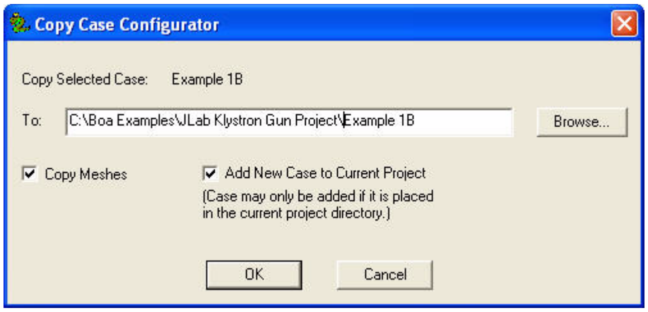


FIGURE 145. *Save Case As...* on the Case sub-

2. Select *Save Case As...* from the sub-menu. The *Copy Case Configurator* dialog will appear.
3. Change the name of the case from ‘Copy of Example 1A’ to ‘Example 1B’ as shown in Figure 146.



Copy Selected Case: Example 1B

To: C:\Boa Examples\JLab Klystron Gun Project\Example 1B

☑ Copy Meshes

☑ Add New Case to Current Project

(Case may only be added if it is placed in the current project directory.)

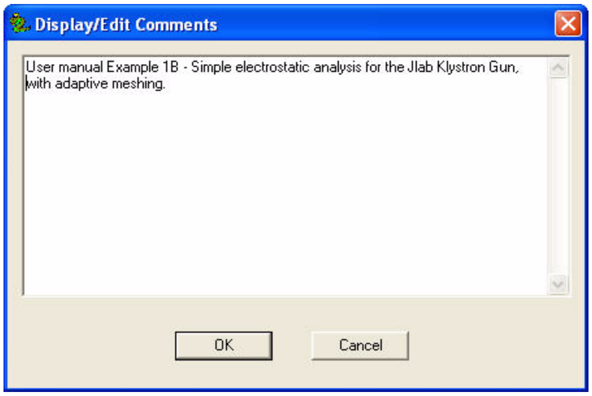
FIGURE 146. Copy Case dialog

4. Leave both checkboxes checked. (The initial mesh for this case is the same as in case ‘Example 1A’ so there is no need to rerun the initial mesh.)

5. Click *OK*. The new case will appear in the Project/Case Manager Tree control and the model will appear in the Main Window.

6. To enter comments for case ‘Example 1B’, right click on the new case (‘Case: Example 1B’) in the Project/Case Manager tree control and select *Display/Edit Comments* from the sub-menu. The *Display/Edit Comments* dialog will appear.

7. Modify the comments as shown in Figure 147. Click *OK* to close the dialog.



User manual Example 1B - Simple electrostatic analysis for the Jlab Klystron Gun, with adaptive meshing.

FIGURE 147. Add New Case dialog

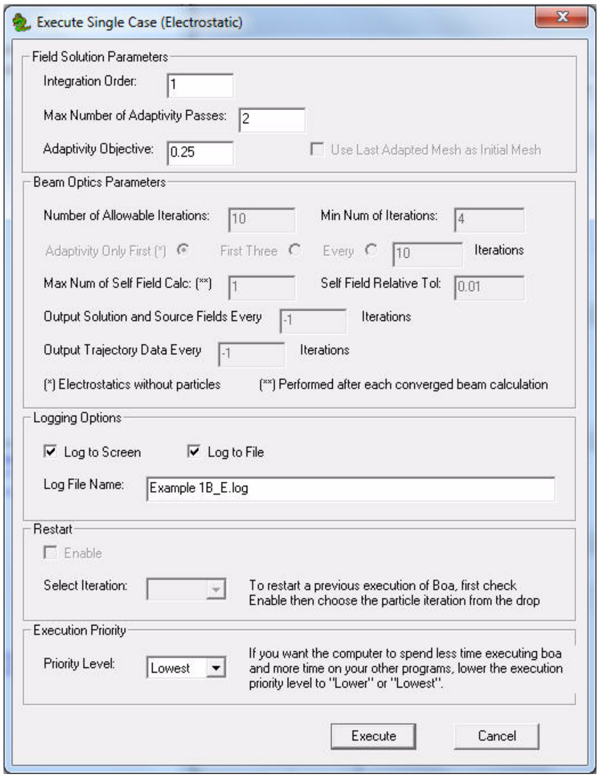
4.1.2.2 Saving a Case

As you continue through this example, it is a good idea to save your work occasionally. To save a case, right click on the *Case* name in the Project/Case Manager tree control and select *Save Case*. (Alternatively, open the *File* pulldown menu and click *Save Case*.)

4.1.2.3 Executing BOA (with Adaptive meshing)

There is no need to run the initial mesh since it is identical to the initial mesh in case ‘Example 1A’ and it has been copied into this case, as discussed above.

1. Open the *Execute* pulldown menu and select *Single Case->Execute*.
2. The *Execute Single Case* dialog shown in Figure 148 will appear.
3. Set *Max Number of Adaptivity Passes* to ‘2’. Leave all other parameters set to the default values. Click *Execute*.
4. A DOS window will appear indicating progress during BOA execution. Because adaptivity is turned on, the runtime will be longer than the first run. When BOA completes, the following text will appear at the bottom of the window “!Proceed to View Results. Press any key to continue...”. Hit any key to close the DOS window and return control back to the user interface.



Execute Single Case (Electrostatic)

Field Solution Parameters

Integration Order: 1

Max Number of Adaptivity Passes: 2

Adaptivity Objective: 0.25

☐ Use Last Adapted Mesh as Initial Mesh

Beam Optics Parameters

Number of Allowable Iterations: 10

Min Num of Iterations: 4

Adaptivity Only First (*) ● First Three ○ Every ○ 10 Iterations

Max Num of Self Field Calc: (**) 1

Self Field Relative Tol: 0.01

Output Solution and Source Fields Every -1 Iterations

Output Trajectory Data Every -1 Iterations

(*) Electrostatics without particles

(**) Performed after each converged beam calculation

Logging Options

☑ Log to Screen ☑ Log to File

Log File Name: Example 1B_E.log

Restart

☐ Enable

Select Iteration:

To restart a previous execution of Boa, first check Enable then choose the particle iteration from the drop

Execution Priority

Priority Level: Lowest

If you want the computer to spend less time executing boa and more time on your other programs, lower the execution priority level to "Lower" or "Lowest".

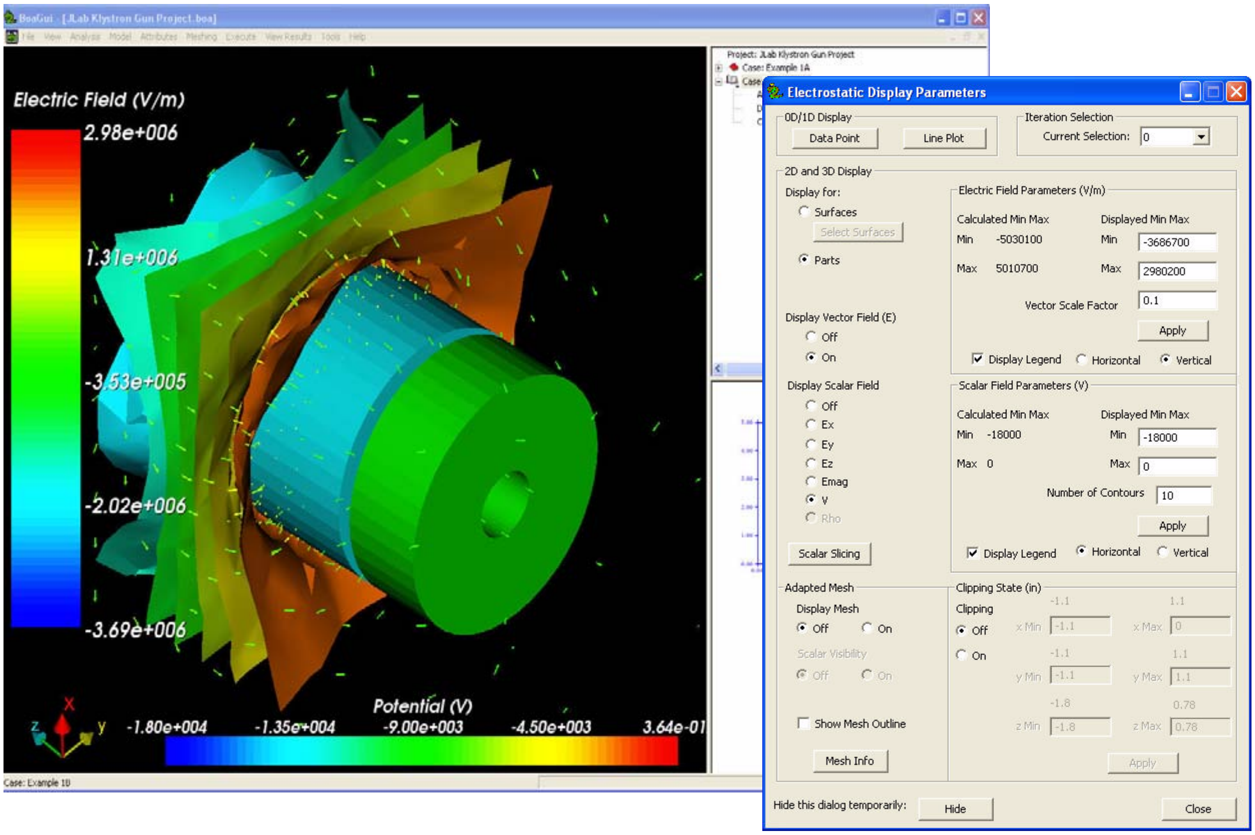
FIGURE 148. Execute dialog for Example 1B

4.1.2.4 Viewing Electrostatic Results (with Adaptive Meshing)

After BOA completes, you must again run the post processor to view the Electrostatic results.

1. Open the *View Results* pulldown menu and select *Electrostatic*. As with the first run, a warning message will indicate that time is required to format the output for plotting. Click ‘Yes’ and wait for the postprocessor to complete.
2. As in Example 1A, the default electrostatic results will appear in the graphics display region and the *Electrostatic Display Parameters* dialog will automatically pop up on top. The results shown correspond to the default settings on the *Electrostatic Display Parameters* dialog (i.e., *Display Vector Field (E)* set to ‘On’ and *Display Scalar Field* set to ‘V’, meaning display the voltage contours). This is shown in Figure 149.

You may notice that the vectors are spaced differently as compared with the vectors shown in Figure 137 (on page 110) for Example 1A. This is due to the difference between the initial mesh and the adapted mesh. (The vectors are located at the mesh corners and the mesh for this example has been adapted).


BoaGui - [JLab Klystron Gun Project.boa]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
Project: JLab Klystron Gun Project
Case: Example 1A
Electric Field (V/m)
2.98e+006
1.31e+006
-3.53e+005
-2.02e+006
-3.69e+006
Potential (V)
-1.80e+004
-1.35e+004
-9.00e+003
-4.50e+003
3.64e-01
x
y
z
Case: Example 1B
Electrostatic Display Parameters
0D/1D Display
Data Point
Line Plot
Iteration Selection
2D and 3D Display
Select Surfaces
Apply
Scalar Slicing
Mesh Info
Hide
Close

Current Selection: 0

Display for: ○ Surfaces ● Parts

Electric Field Parameters (V/m)

	Calculated Min Max		Displayed Min Max
Min	-5030100	Min	-3686700
Max	5010700	Max	2980200

Vector Scale Factor 0.1

☑ Display Legend ○ Horizontal ● Vertical

Display Vector Field (E): ○ Off ● On

Display Scalar Field: ○ Off ○ Ex ○ Ey ○ Ez ○ Emag ● V ○ Rho

Scalar Field Parameters (V)

	Calculated Min Max		Displayed Min Max
Min	-18000	Min	-18000
Max	0	Max	0

Number of Contours 10

☑ Display Legend ● Horizontal ○ Vertical

Adapted Mesh

Display Mesh: ● Off ○ On

Scalar Visibility: ● Off ○ On

☐ Show Mesh Outline

Clipping State (in)

Clipping: ● Off ○ On

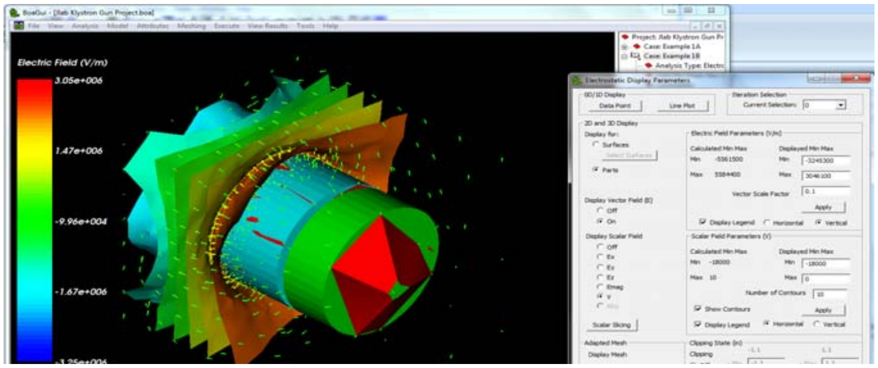
x Min -1.1 x Max 0 (-1.1 1.1)

y Min -1.1 y Max 1.1 (-1.1 1.1)

z Min -1.8 z Max 0.78 (-1.8 0.78)

Hide this dialog temporarily:

FIGURE 149. Electrostatic results with Voltage Contours On, Vector Field On (executed with *Max Number of Adaptivity Passes* = 2) UPDATE THIS PICTURE?. In my recheck with new version this picture looks different (see below). Confirm with Thuc that new picture is correct...


BoaGui - [Jlab Klystron Gun Project.boa]
Electric Field (V/m)
3.05e+006
1.47e+006
-9.96e+004
-1.67e+006
Project: Jlab Klystron Gun Pr
Case: Example 1A
Case: Example 1B
Electrostatic Display Parameters

[Mike this is where I stopped my updates when I sent to you on 11/18/09]

4.1.2.4.1 Viewing the Adapted Mesh

Next, look at the adapted mesh to see how it differs from the initial mesh.

1. The *Electrostatic Display Parameters* dialog should still be open. In the *Adapted Mesh* section, change the *Display Mesh* radio button to 'On'. The mesh will appear as shown in Figure 150. Notice that the adapted mesh is coarser towards the front of the view and finer towards the back.
2. To see the inside of the mesh, change the *Clipping* radio button to 'On' in the *Clipping State* section. Set *x Max* to '0' and set *yMin* back to '-1.1'.
3. In order to see inside the gun assembly, hide the anode and the mod anode as follows. Open the *Attributes* menu. The *Attributes Assignment Properties* dialog will pop up on top of the *Electrostatic Display Parameters* dialog. Select mod anode and uncheck the *Visible* checkbox. Do the same for the anode.

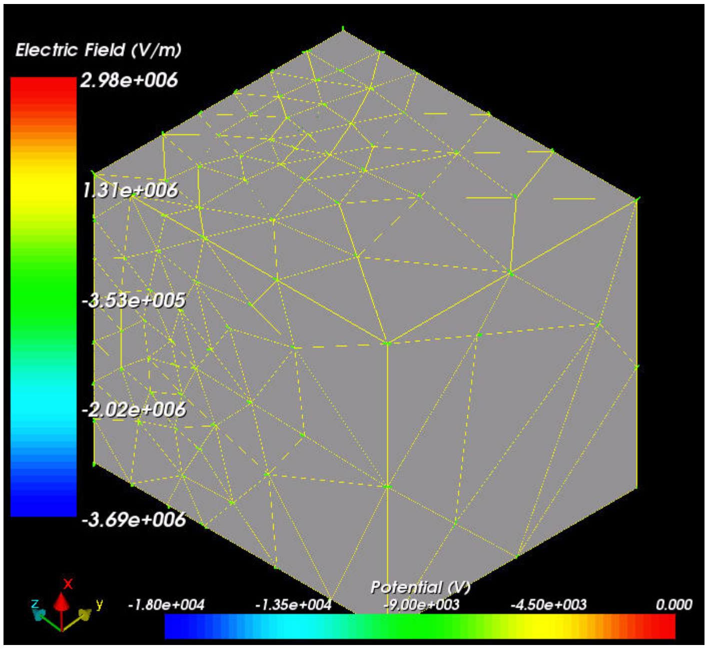

Electric Field (V/m)
2.98e+006
1.31e+006
-3.53e+005
-2.02e+006
-3.69e+006
x
z
y
-1.80e+004
-1.35e+004
Potential (V)
-9.00e+003
-4.50e+003
0.000

FIGURE 150. Adapted Mesh On

4. Click *Ok* to close the *Attributes Assignment Properties* dialog.
5. The mesh will appear similar to that shown in Figure 151. Use the mouse to move the picture around and view it from different perspectives.Notice how the mesh is coarser in some areas and finer in the areas of interest.

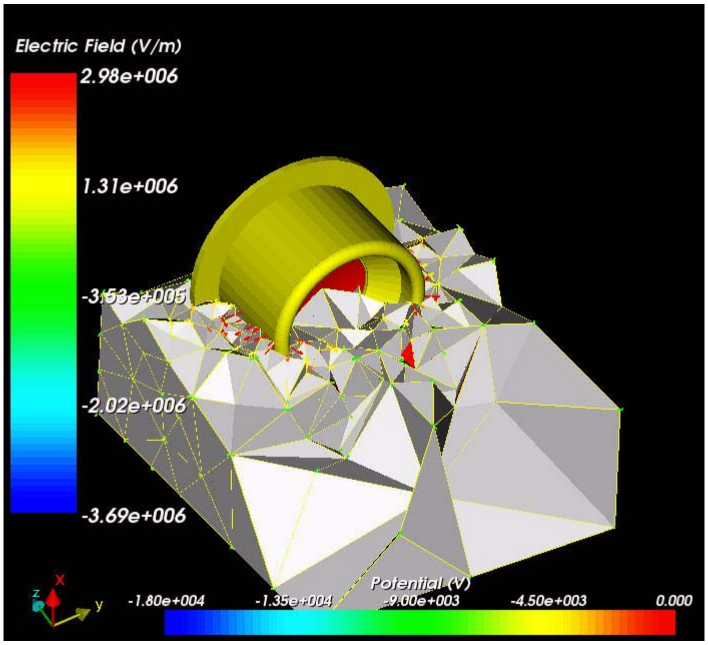
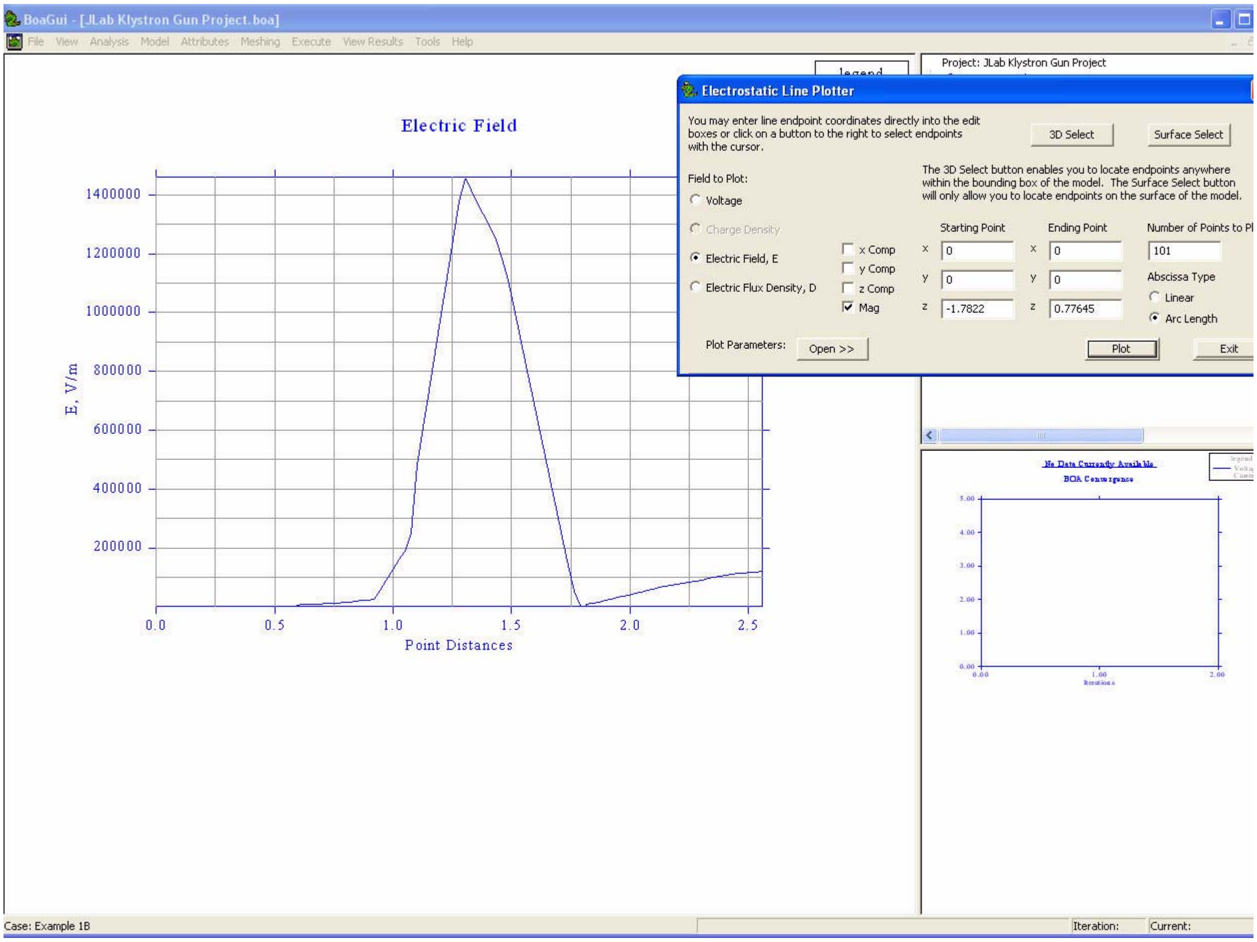

Electric Field (V/m)
2.98e+006
1.31e+006
-3.53e+005
-2.02e+006
-3.69e+006
x
z
y
-1.80e+004
-1.35e+004
Potential (V)
-9.00e+003
-4.50e+003
0.000

FIGURE 151. Adapted Mesh On, Clipping State On

4.1.2.4.2 Line plot of the E magnitude along the beam axis (i.e., z axis)

Figure 152 shows a line plot of the E magnitude along the z (or beam) axis. To generate this picture, perform the following steps.


BoaGui - [JLab Klystron Gun Project.boa]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
Electric Field
legend
1400000
1200000
1000000
800000
600000
400000
200000
E, V/m
0.0
0.5
1.0
1.5
2.0
2.5
Point Distances
Electrostatic Line Plotter
3D Select
Surface Select
Field to Plot:
Voltage
Charge Density
Electric Field, E
Electric Flux Density, D
x Comp
y Comp
z Comp
Mag
Starting Point
Ending Point
Number of Points to Pl
Abscissa Type
Linear
Arc Length
Plot Parameters:
Open >>
Plot
Exit
No Data Currently Available
BOA Convergence
Iterations
Iteration:
Current:

Project: JLab Klystron Gun Project

You may enter line endpoint coordinates directly into the edit boxes or click on a button to the right to select endpoints with the cursor.

The 3D Select button enables you to locate endpoints anywhere within the bounding box of the model. The Surface Select button will only allow you to locate endpoints on the surface of the model.

Field to Plot: ● Electric Field, E; ☑ Mag

Starting Point: x 0, y 0, z -1.7822

Ending Point: x 0, y 0, z 0.77645

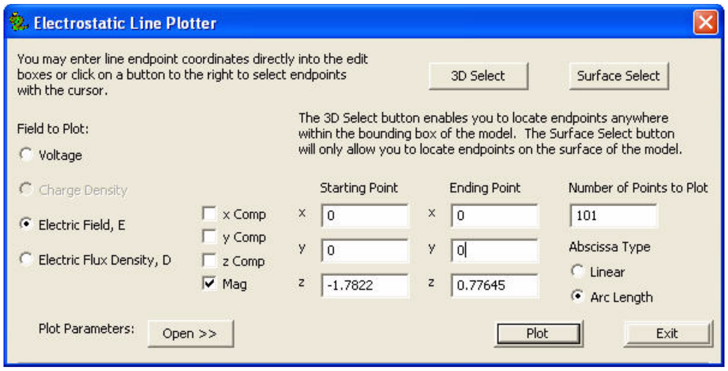
Number of Points: 101

Abscissa Type: ● Arc Length

Case: Example 1B

FIGURE 152. Line plot of E magnitude along beam axis (i.e., z axis) for example 1B

1. The *Electrostatic Display Parameters* dialog should still be open.
2. Click on the *Line Plot* located in the *0D/1D section* at the top of the dialog. The *Electrostatic line Plotter* dialog will appear.
3. Change the *Field to Plot* radio button to *Electric Field, E* and leave the *Mag* checkbox checked.
4. Set the *Starting Point* and *Ending point* for both *x* and *y* to ‘0’. Leave the *Starting Point* and *Ending Point* for *z* at the default.
5. Click *Plot*. The line plot will appear in the main window, as shown in Figure 152.


Electrostatic Line Plotter
3D Select
Surface Select
Field to Plot:
○ Voltage
○ Charge Density
● Electric Field, E
○ Electric Flux Density, D
☐ x Comp
☐ y Comp
☐ z Comp
☑ Mag
Starting Point
Ending Point
Number of Points to Plot
Abscissa Type
○ Linear
● Arc Length
Plot Parameters:
Open >>
Plot
Exit

You may enter line endpoint coordinates directly into the edit boxes or click on a button to the right to select endpoints with the cursor.

The 3D Select button enables you to locate endpoints anywhere within the bounding box of the model. The Surface Select button will only allow you to locate endpoints on the surface of the model.

Starting Point: x 0, y 0, z -1.7822

Ending Point: x 0, y 0, z 0.77645

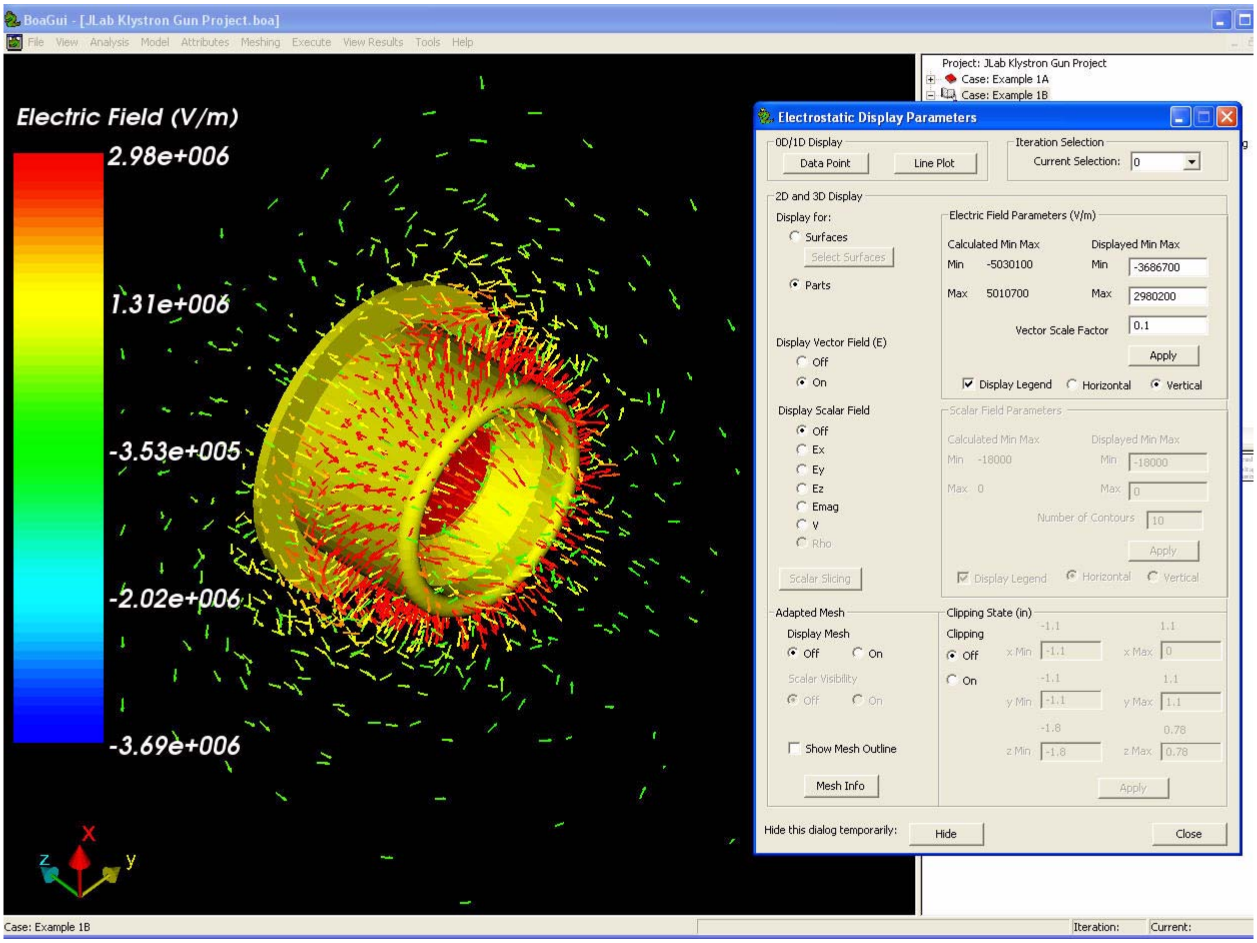
Number of Points to Plot: 101

FIGURE 153. Electrostatic Line Plotter

6. Compare this graph with the Line Plot of the E magnitude you created for Example 1A in Figure 143 (on page 115). Notice that the peak field with adaptivity (Figure 152) is higher and smoother at both bottoms of the bell curve than the peak field without adaptivity (Figure 143).

4.1.2.4.3 Viewing additional results

As in example 1A, you may want to experiment with different parameters to see different results. For example, to get the result shown in Figure 154 follow the steps below.


BoaGui - [JLab Klystron Gun Project.boa]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
Electric Field (V/m)
2.98e+006
1.31e+006
-3.53e+005
-2.02e+006
-3.69e+006
x
y
z
Project: JLab Klystron Gun Project
Case: Example 1A
Case: Example 1B
Electrostatic Display Parameters
Data Point
Line Plot
Select Surfaces
Scalar Slicing
Mesh Info
Apply
Hide
Close
Iteration:
Current:

Case: Example 1B

0D/1D Display

Iteration Selection: Current Selection: 0

2D and 3D Display

Display for: ○ Surfaces ● Parts

Electric Field Parameters (V/m)

Calculated Min Max		Displayed Min Max	
Min	-5030100	Min	-3686700
Max	5010700	Max	2980200

Vector Scale Factor: 0.1

☑ Display Legend ○ Horizontal ● Vertical

Display Vector Field (E): ○ Off ● On

Display Scalar Field: ● Off ○ Ex ○ Ey ○ Ez ○ Emag ○ V ○ Rho

Scalar Field Parameters

Calculated Min Max		Displayed Min Max	
Min	-18000	Min	-18000
Max	0	Max	0

Number of Contours: 10

☑ Display Legend ● Horizontal ○ Vertical

Adapted Mesh

Display Mesh: ● Off ○ On

Scalar Visibility: ● Off ○ On

☐ Show Mesh Outline

Clipping State (in)

Clipping: ● Off ○ On

x Min	-1.1	x Max	0
y Min	-1.1	y Max	1.1
z Min	-1.8	z Max	0.78

Hide this dialog temporarily:

FIGURE 154. E field ‘On’ with anode and mod anode hidden

1. Set the *Clipping State* radio button to ‘Off’.
2. Set the *Adapted Mesh* radio button to ‘Off’.
3. Set the *Display Scalar Field* radio button to ‘Off’.
4. Leave the *Display Vector Field (E)* radio button set to ‘On’.
5. When you are done viewing the results, continue on to Example 1C. Make sure to save the case. (Right-click on the case name in the Project/Case Tree Control and select *Save Case*.)
6. Close the Case. (Right-click on the case name in the Project/Case Tree Control and select *Close Case*.)

4.1.3 Example 1C - Electrostatic Beam Analysis (with Thermionic Emitter)

In this example, you will extend Example 1A to create a beam optics analysis. Adaptivity is not used in this example.

4.1.3.1 Creating a new Case

To begin, you will copy the first Case, Example 1A, to a new Case, as follows.

1. Right click on the ‘Case: Example 1A’ case in the Project/Case Manager tree control. The submenu shown in Figure 155 will appear.
2. Select *Save Case As...* from the sub-menu. The *Copy Case Configurator* dialog will appear.

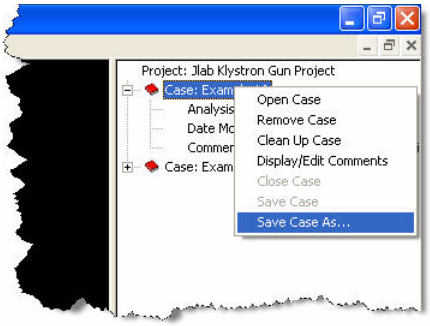
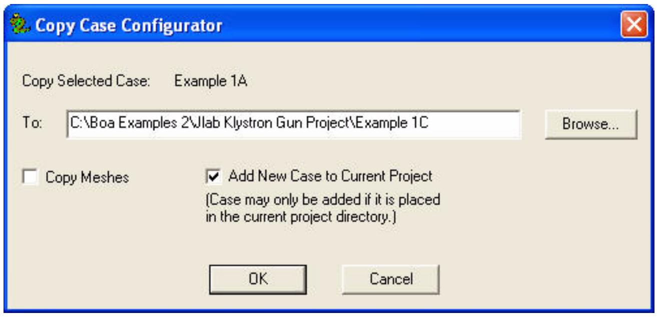

Project: Jlab Klystron Gun Project
Case: Exam
Analysis
Date Mo
Commer
Case: Exam
Open Case
Remove Case
Clean Up Case
Display/Edit Comments
Close Case
Save Case
Save Case As...

FIGURE 155. ***Save Case As...*** **on the Case sub-menu**

3. Change the name of the Case from ‘Copy of Example 1A’ to ‘Example 1C’ as shown in Figure 156.


Copy Case Configurator
Browse...
OK
Cancel

Copy Selected Case: Example 1A

To: C:\Boa Examples 2\Jlab Klystron Gun Project\Example 1C

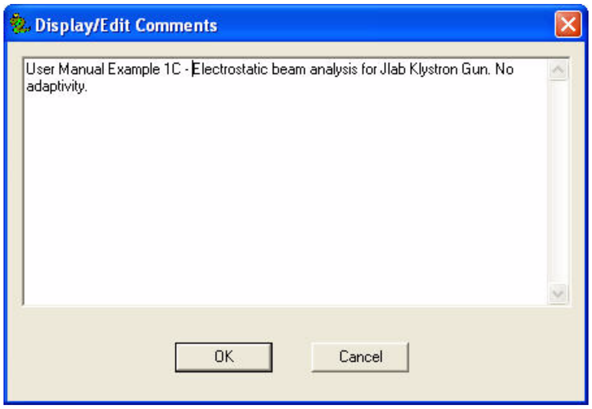
☐ Copy Meshes

☑ Add New Case to Current Project

(Case may only be added if it is placed in the current project directory.)

FIGURE 156. Copy Case dialog

4. Uncheck the *Copy Meshes* checkbox. (You will generate a different initial mesh for this case.) Leave the other checkbox checked.
5. Click *OK*. The new case will appear in the Project/Case Manager tree control and the model will appear in the graphics display region.
6. To add comments for Example 1C, right click on the new case (‘Case: Example 1C’) in the Project/Case Manager tree control. Select *Display/Edit Comments* from the sub-menu. The *Display/Edit Comments* dialog will appear.
7. Modify the comments as shown in Figure 157.
8. Click *OK* to close the dialog.


Display/Edit Comments
OK
Cancel

User Manual Example 1C - Electrostatic beam analysis for Jlab Klystron Gun. No adaptivity.

FIGURE 157. Display/Edit Comments dialog

4.1.3.2 Saving a Case

As you continue through this example, it is a good idea to save your work occasionally. To save a case, right click on the *Case* name in the Project/Case Manager tree control and select *Save Case*. (Alternatively, open the *File* pull-down menu and click *Save Case*.)

4.1.3.3 Defining Analysis Type

Example 1C is an *Electrostatic Beam* analysis type.

1. Open the *Analysis* pulldown menu and select *Electrostatic Beam* analysis type.

4.1.3.4 Assigning Beam Optics Attributes

Because this example is based on Example 1A, the geometry file has already been specified and the electrostatic attributes have already been assigned. Now you will assign the beam optics attributes.

1. Open the *Attribute Assignment Properties* dialog. (Click on the *Attributes* menu item.) In the next few steps, the goal will be to define the surface of the cathode as a thermionic emitter. In order to better see the cathode, make sure the Anode and the Mod Anode are invisible by unchecking the *Visible* checkbox for both.
2. Rotate the model so that the cathode surface is clearly visible as shown in Figure 158. To center the visible model, open the *View* menu and click *Refresh View.*

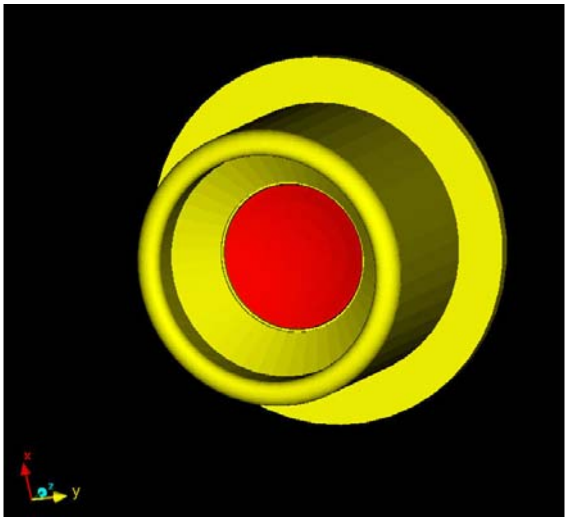
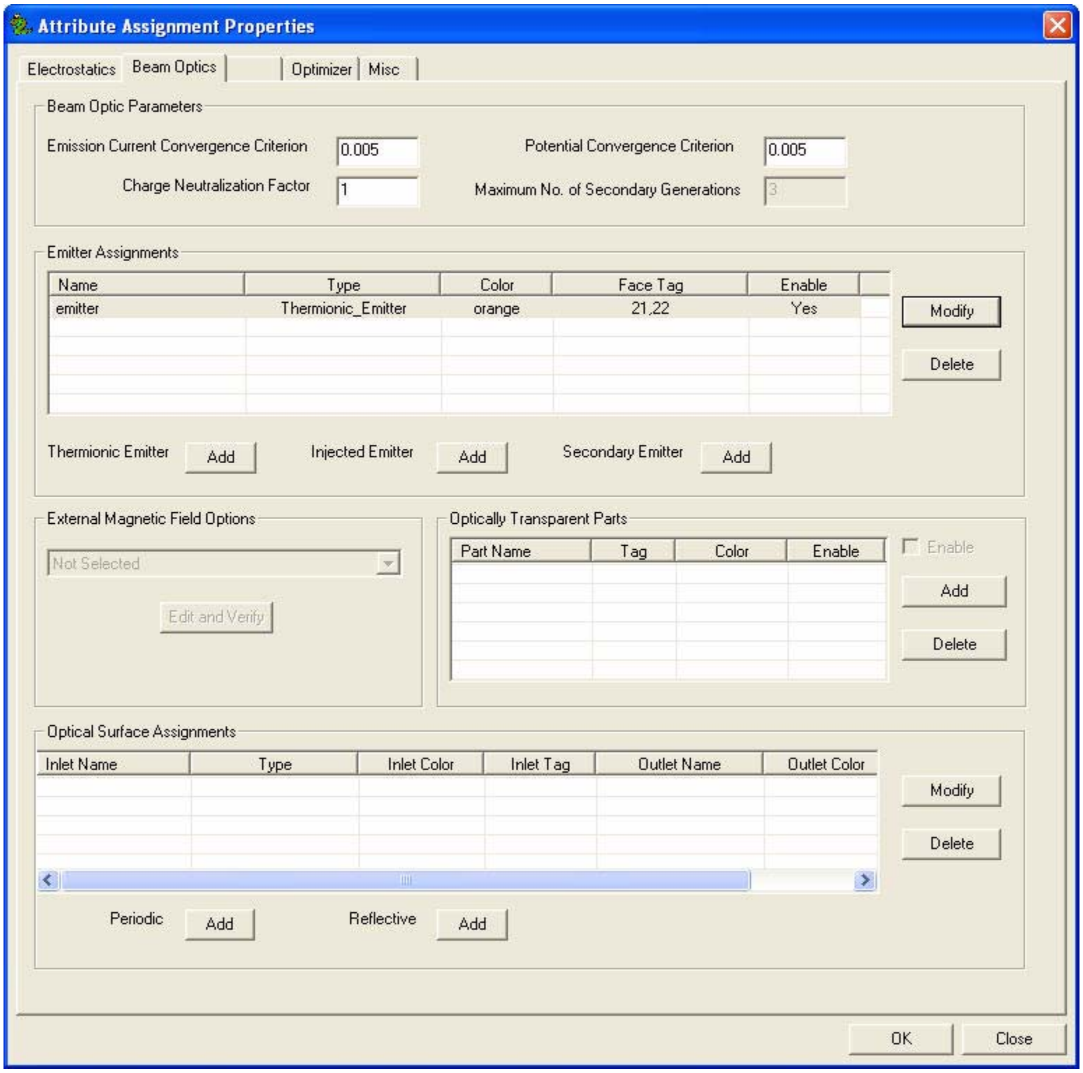


FIGURE 158. Geometry rotated so that cathode emitter surface is visible.

3. Click on the *Beam Optics* tab. The dialog shown in Figure 159 will appear. Unlike Figure 159, however, the *Emitter Assignments* table will be blank.


Attribute Assignment Properties
Electrostatics
Beam Optics
Optimizer
Misc
Beam Optic Parameters
Emitter Assignments
Modify
Delete
Thermionic Emitter
Add
Injected Emitter
Add
Secondary Emitter
Add
External Magnetic Field Options
Not Selected
Edit and Verify
Optically Transparent Parts
Part Name
Tag
Color
Enable
Enable
Add
Delete
Optical Surface Assignments
Inlet Name
Type
Inlet Color
Inlet Tag
Outlet Name
Outlet Color
Modify
Delete
Periodic
Add
Reflective
Add
OK
Close

Emission Current Convergence Criterion 0.005

Potential Convergence Criterion 0.005

Charge Neutralization Factor 1

Maximum No. of Secondary Generations 3

Name	Type	Color	Face Tag	Enable
emitter	Thermionic_Emitter	orange	21,22	Yes

FIGURE 159. Attribute Assignment dialog box, Beam Optics tab

4. Leave the *Beam Optic Parameters* set to the defaults.
5. In the *Emitter Assignments* section, click the *Thermionic Emitter Add* button beneath the table. This will bring up the graphics display window containing the electrostatic geometry model.

6. Select the cathode surface by pressing Ctrl-Left mouse button. The cathode surface will appear selected (it will change color) as shown in Figure 160. Click enter to complete the selection.
7. The dialog box shown in Figure 161 will appear. In this case, the surface you selected corresponds to a part surface named ‘emitter’ in the geometry file. Therefore the *Name* box automatically shows the name ‘emitter’ and it is not editable. (If the selected surface did not correspond to a named part surface in the geometry model, this field would be editable.)
8. Set the *Color* dropdown box to ‘orange’.
9. Set *Approximate Number of Particle Launch Sites* to ‘100’.

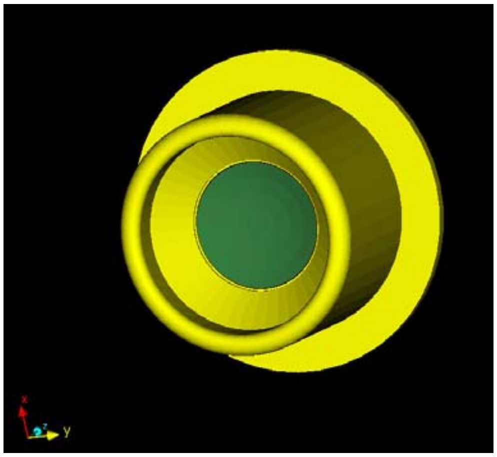
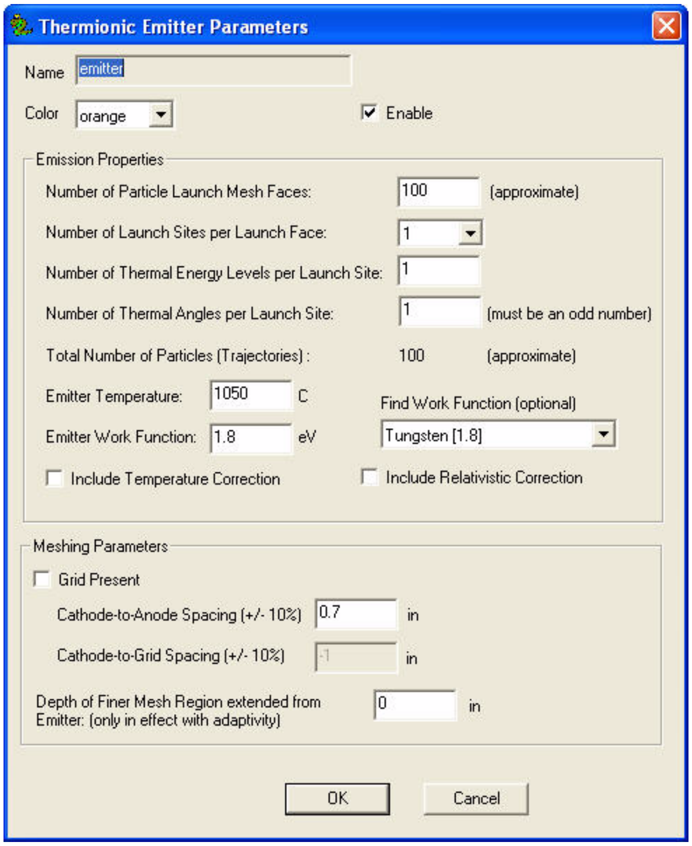


FIGURE 160. Defining the cathode

10. In the *Meshing Parameters (for Beam Analysis)* box, define the cathode to anode spacing by entering 0.7 in the *Cathode-to-Anode Spacing* data entry box. (Leave the *Grid Present* checkbox unchecked.)
11. Click *OK* to complete the emitter definition. The newly defined emitter will appear in the *Emitter Assignments* table as shown in Figure 159.
12. Click the *Electrostatic* tab. Check the *Visible* checkbox for the Anode and the Mod Anode to make them visible again.
13. Click *OK* to close the *Attribute Assignment* dialog.
14. To reset the model position, open the *View* menu and select *Reset View.*

4.1.3.5 Generating the Initial Electrostatic Mesh

Once you complete the attribute assignments, you are ready to generate the initial electrostatic mesh. Unlike in Examples 1A and 1B, you will define areas of finer mesh control.



Thermionic Emitter Parameters

Name: emitter

Color: orange

☑ Enable

Emission Properties

Number of Particle Launch Mesh Faces: 100 (approximate)

Number of Launch Sites per Launch Face: 1

Number of Thermal Energy Levels per Launch Site: 1

Number of Thermal Angles per Launch Site: 1 (must be an odd number)

Total Number of Particles (Trajectories): 100 (approximate)

Emitter Temperature: 1050 C

Emitter Work Function: 1.8 eV

Find Work Function (optional): Tungsten [1.8]

☐ Include Temperature Correction

☐ Include Relativistic Correction

Meshing Parameters

☐ Grid Present

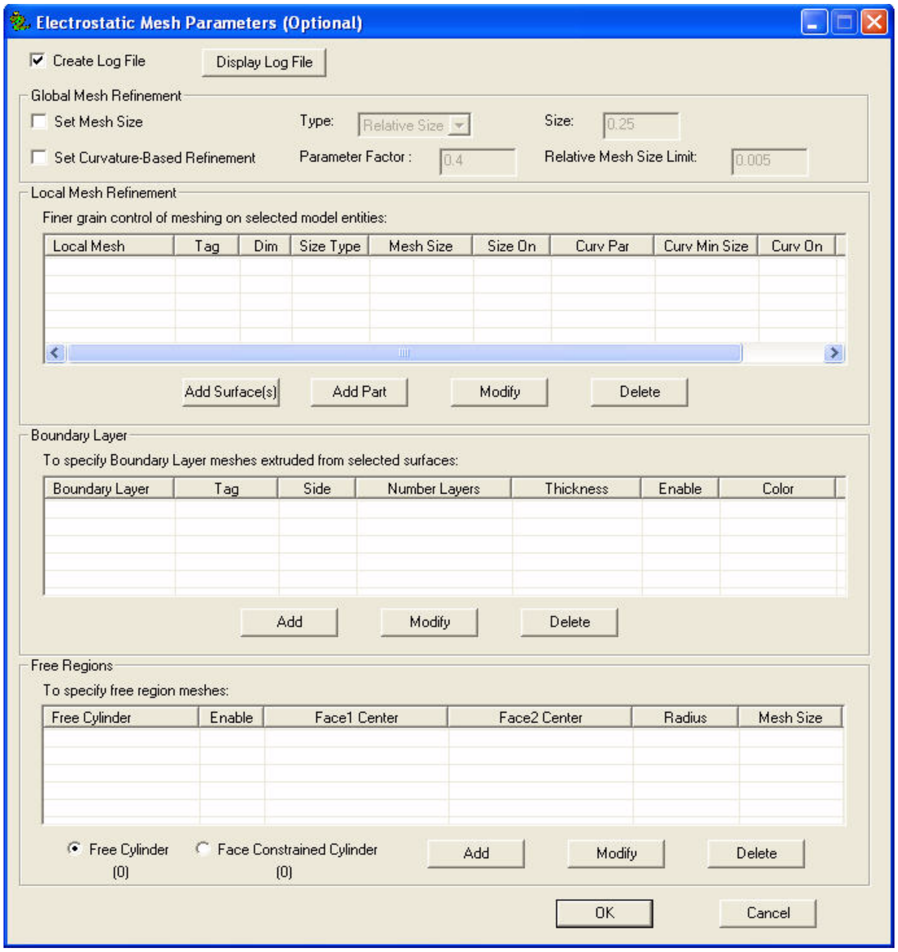
Cathode-to-Anode Spacing (+/- 10%) 0.7 in

Cathode-to-Grid Spacing (+/- 10%) -1 in

Depth of Finer Mesh Region extended from Emitter: (only in effect with adaptivity) 0 in

FIGURE 161. Defining a thermionic emitter

1. Open the *Meshing* pulldown menu and select *Parameters*. The dialog box shown in Figure 162 will appear.


Electrostatic Mesh Parameters (Optional)
Display Log File
Global Mesh Refinement
Local Mesh Refinement
Local Mesh
Tag
Dim
Size Type
Mesh Size
Size On
Curv Par
Curv Min Size
Curv On
Add Surface(s)
Add Part
Modify
Delete
Boundary Layer
Boundary Layer
Tag
Side
Number Layers
Thickness
Enable
Color
Add
Modify
Delete
Free Regions
Free Cylinder
Enable
Face1 Center
Face2 Center
Radius
Mesh Size
Add
Modify
Delete
OK
Cancel

☑ Create Log File

☐ Set Mesh Size Type: Relative Size Size: 0.25

☐ Set Curvature-Based Refinement Parameter Factor : 0.4 Relative Mesh Size Limit: 0.005

Finer grain control of meshing on selected model entities:

To specify Boundary Layer meshes extruded from selected surfaces:

To specify free region meshes:

● Free Cylinder (0) ○ Face Constrained Cylinder (0)

FIGURE 162. Meshing Parameters Dialog Box

2. All tables in this section will be blank initially. In this example, the goal is to define the local mesh density (i.e., *Local Mesh Refinements*) for three different sets of surfaces. These include the tunnel of the anode, a portion of the focus electrode and the tunnel of the mod anode. (Defining a *Local Mesh Refinement* means that you specify the mesh for either a set of surfaces or for a part. The initial mesh generation will create these meshes closely to your specifications. Furthermore, when adaptation is requested, the adaptation routine may refine *but never coarsen* these meshes.)
3. Click the *Add Surface(s)* button beneath the table in the *Local Mesh Refinement* section. This will bring up the graphics display window containing the electrostatic geometry model.

4. Confirm that the model is oriented so that the Anode is in the front as shown in Figure 163.

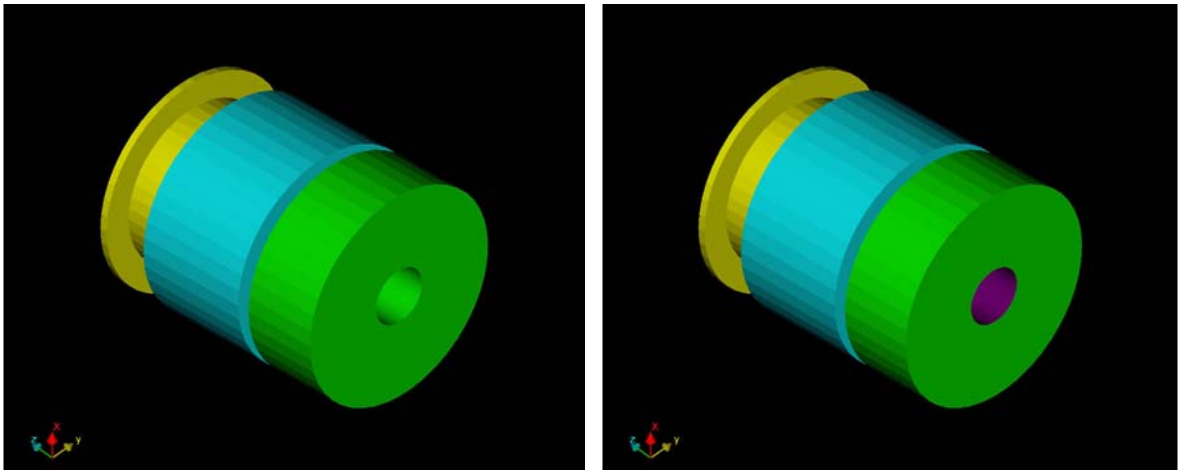
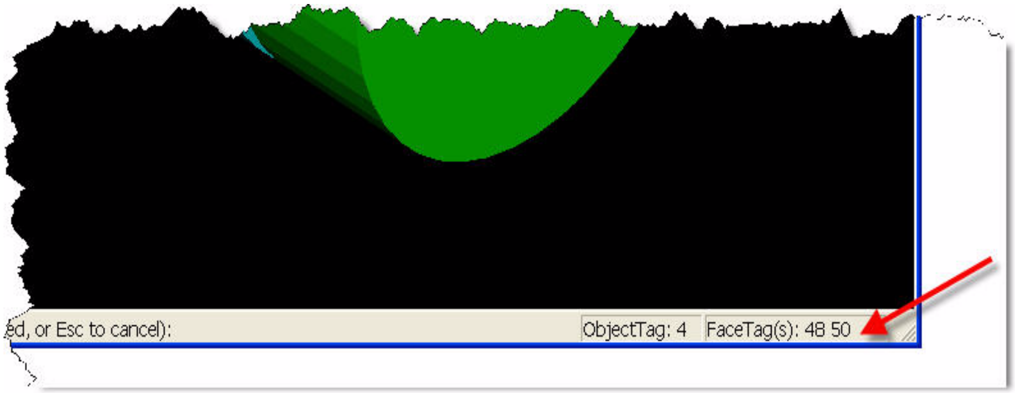


FIGURE 163. The Anode Tunnel (Left image is prior to defining the AnodeTunnel model surface. Right image is after defining the AnodeTunnel model surface.)

The goal is to define the portion shown in purple in the right-hand image (that is, the inner tunnel of the anode) as a *Local Mesh Refinement* (i.e., as a mesh controlled surface). Use the Ctrl-Left mouse button to select the tunnel. The inside of the anode will appear selected (it will change color) and the status bar on the bottom of the screen will display the selected surface tags (48 and 50), as shown in Figure 164. (The internal algorithms used by BOA separate each part in the geometry model into multiple ‘surface tags’. In order to specify a desired set of surfaces in BoaGui, you usually have to make a separate selection for each surface tag. However, if the geometry file contains a ‘named surface’ (an optional feature in some CAD programs), BoaGui allows you to select all of the surface tags included in the named surface as one single entity. In this case, the JLAB Gun.SAT file contains the named surface ‘AnodeTunnel’ and it consists of surface tags 48 and 50. Because it is a named surface in the geometry file, you only need to select one of the tags; all tags associated with the named surface will automatically get included in the selection.) Click *Enter* to complete the selection.



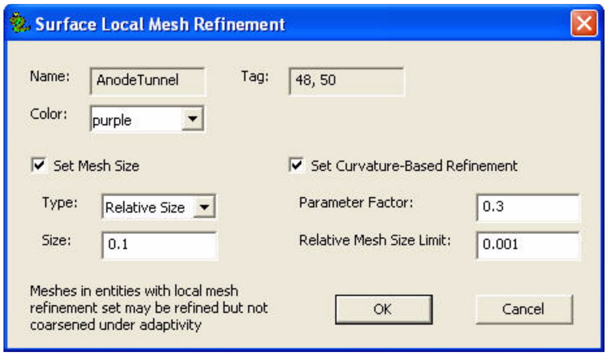
ed, or Esc to cancel):

ObjectTag: 4

FaceTag(s): 48 50

FIGURE 164. The Surface Tag IDs on the Status

5. The dialog in Figure 165 will appear. The *Name* entry box will not be editable. It will be prepopulated with the name ‘AnodeTunnel’. This is the name provided in the JLAB Gun.SAT file for this set of surface tags. Set the *Color* dropdown box to ‘purple’. Check both the *Set Mesh Size* and *Set Curvature-Based Refinement boxes*. Leave the remaining entries at their default values. Click *OK* to complete the surface definition.

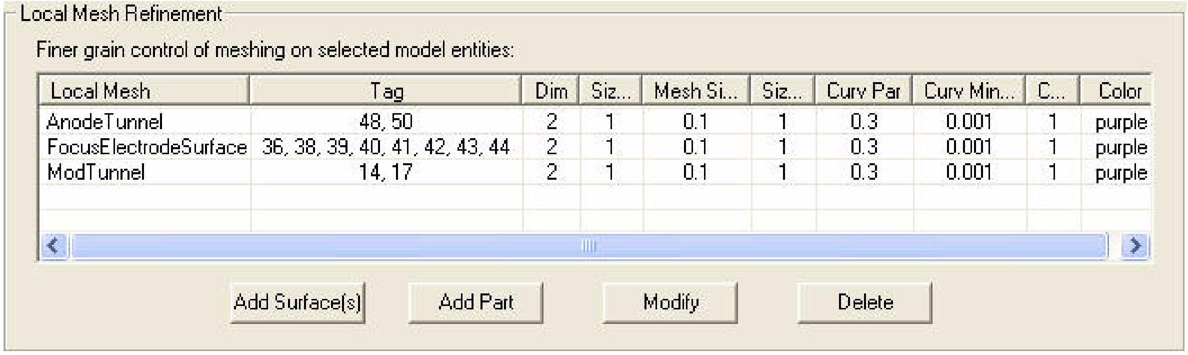

Surface Local Mesh Refinement

Name: AnodeTunnel
Tag: 48, 50
Color: purple
☑ Set Mesh Size
Type: Relative Size
Size: 0.1
☑ Set Curvature-Based Refinement
Parameter Factor: 0.3
Relative Mesh Size Limit: 0.001

Meshes in entities with local mesh refinement set may be refined but not coarsened under adaptivity

FIGURE 165. Defining a Local Mesh Refinement for the Surface ‘Anode Tunnel’

6. The newly defined surface will appear in the *Local Mesh Refinement* section, as shown in the first entry of the table in Figure 166.


Local Mesh Refinement

Finer grain control of meshing on selected model entities:

Local Mesh	Tag	Dim	Siz...	Mesh Si...	Siz...	Curv Par	Curv Min...	C...	Color
AnodeTunnel	48, 50	2	1	0.1	1	0.3	0.001	1	purple
FocusElectrodeSurface	36, 38, 39, 40, 41, 42, 43, 44	2	1	0.1	1	0.3	0.001	1	purple
ModTunnel	14, 17	2	1	0.1	1	0.3	0.001	1	purple

FIGURE 166. Local Mesh Refinements

7. The next surface to define is part of the focus electrode. It will be easier to select if only the focus electrode is visible. Click the *Attributes* menu item to bring up the *Attribute Assignment Properties* dialog. (Notice that you can open this dialog without closing the *Electrostatic Mesh Parameters (Optional)* dialog. The *Attribute Assignment Properties* dialog simply pops up on top.) In the *Properties at the Part Level* table on the *Electrostatics* tab, make all parts invisible except the Focus Electrode. That is, select the Mod Anode and uncheck the *Visible* checkbox. Do the same for the Cathode and the Anode. Click *OK* to close the *Attribute Assignment Properties* dialog.
8. The *Electrostatic Mesh Parameters (Optional)* dialog will still be open. In the *Local Mesh Refinement* section, click the *Add Surface(s)* button beneath the table. This will bring up the graphics display window containing the electrostatic geometry model.
9. Orient the model as shown in Figure 167. The goal is to define the portion shown in purple in the right-hand image as a *Local Mesh Refinement* (i.e., as a mesh controlled surface). Use the Ctrl-Left mouse button to make the necessary selections. You will need to move the model around in order to select all associated surface tags. (The internal algorithms used by BOA have broken this portion of the Focus Electrode part into eight separate surface tags: 36, 38, 39,

40, 41, 42, 43, 44. Unlike with the selection of the Anode Tunnel in a the previous surface definition, the JLAB Gun.SAT file does not contain a named surface for this area. Therefore you must make a selection for each of the eight surface tags separately. As you make each selection, the surface tag numbers will appear on the status bar.)

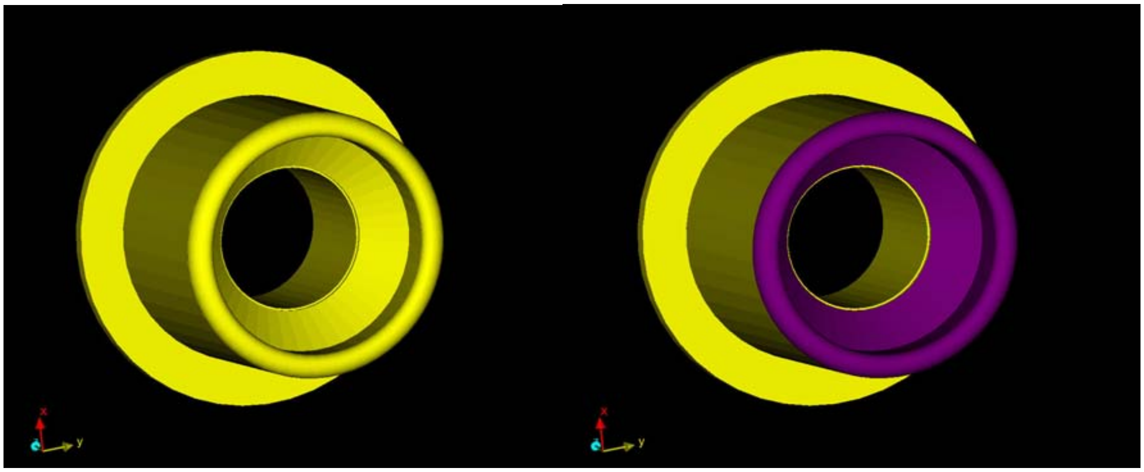
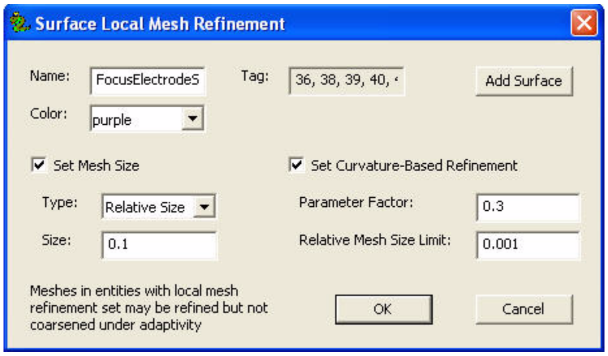


FIGURE 167. The Focus Electrode. (Left image is prior to defining the FocusElectrode surface. Right image is after defining the FocusElectrode surface.)

a. To begin, Select *Add Surface(s)* below the table in the *Local Mesh Refinement* section. The dialog box will disappear exposing the model window. Ctrl-left click on any region indicated in purple in Figure 167, then hit Enter. The dialog shown in Figure 168 will appear. Enter *FocusElectrodeSurface* (without a space) in the *Name* box. Check the *Set Mesh Size* checkbox and the *Set Curvature-Based Refinement* checkbox and leave the associated entries at their default values.



Surface Local Mesh Refinement

Name: FocusElectrodeS

Tag: 36, 38, 39, 40,

Color: purple

☑ Set Mesh Size

Type: Relative Size

Size: 0.1

☑ Set Curvature-Based Refinement

Parameter Factor: 0.3

Relative Mesh Size Limit: 0.001

Meshes in entities with local mesh refinement set may be refined but not coarsened under adaptivity

FIGURE 168. Defining a Local Mesh Refinement for the Surface ‘FocusElectrodeSurface’

b. Click on *Add Surface* in the top right of the box. This will close the dialog box and return to the model window. Select another surface from the purple region shown in Figure 167. (You may need to move the *Electrostatic Mesh* parameters dialog to access the region.) Hit the Enter key to return to the *Surface Local Mesh Refinement* dialog and note that another surface tag was added to the *Tag* box.

c. Repeat this procedure (*Add Surface* > select surface > Enter > *Add Surface*) until all purple regions shown in Figure 167 are selected. If you lose track of which surfaces have been selected, close the *Surface Local Mesh Refinement* dialog box and select FocusElectrodeSurface in the *Local Mesh Refinement* table. This will indicate by color which surfaces have been chosen. Select the *Modify* button to add any missed surfaces. (As you can see, selecting surfaces that have not been named in the CAD file can be tedious. For this reason, among others, it is highly recommended that you name all important surfaces in your CAD file prior to importing it into BOAGui.)

d. Once you have completed the ‘FocusElectrodeSurface’ definition, it will appear in the *Local Mesh Refinement* section, as shown in the second entry of the table in Figure 166.

5. The next surface to define is on the Mod Anode. It will be easier to select if you make only the Mod Anode visible. Click the *Attributes* menu item to bring up the *Attribute Assignment Properties* dialog. In the *Properties at the Part Level* table on the *Electrostatics* tab, select the Focus Electrode and uncheck the *Visible* checkbox, then select the Mod Anode and check the *Visible* checkbox. Click *OK* to close the *Attribute Assignment Properties* dialog.
6. The *Electrostatic Mesh Parameters (Optional)* dialog will still be open. In the *Local Mesh Refinement* section, click the *Add Surface(s)* button beneath the table. This will bring up the graphics display window containing the electrostatic geometry model.
7. Turn the model so that the Mod Anode is oriented as shown in Figure 169. The goal is to control the mesh density on the portion shown in purple in the picture on the right. As before, select *Add Surface(s)* in the *Local Mesh Refinement* region and use the Ctrl-Left mouse button to select one of the surfaces. This will select a named surface with two tags (surface tags 14 and 17).

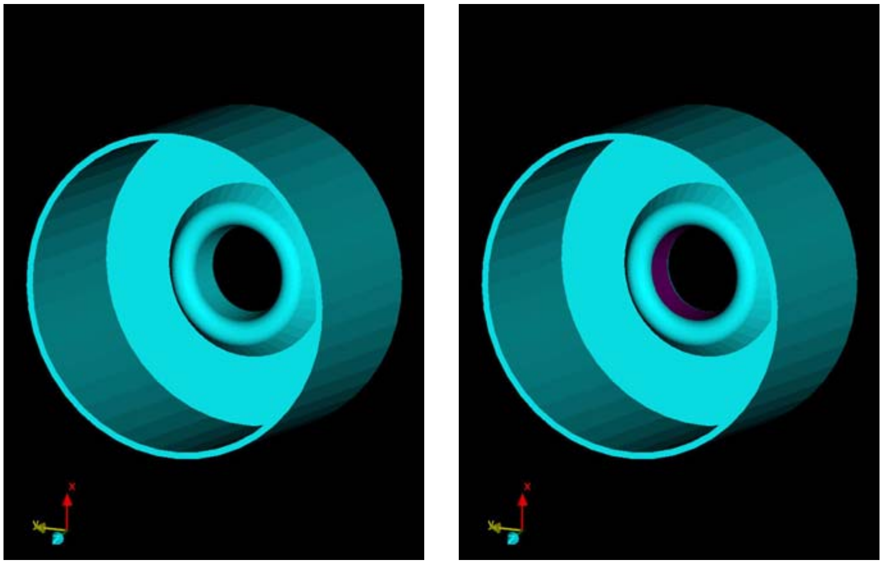


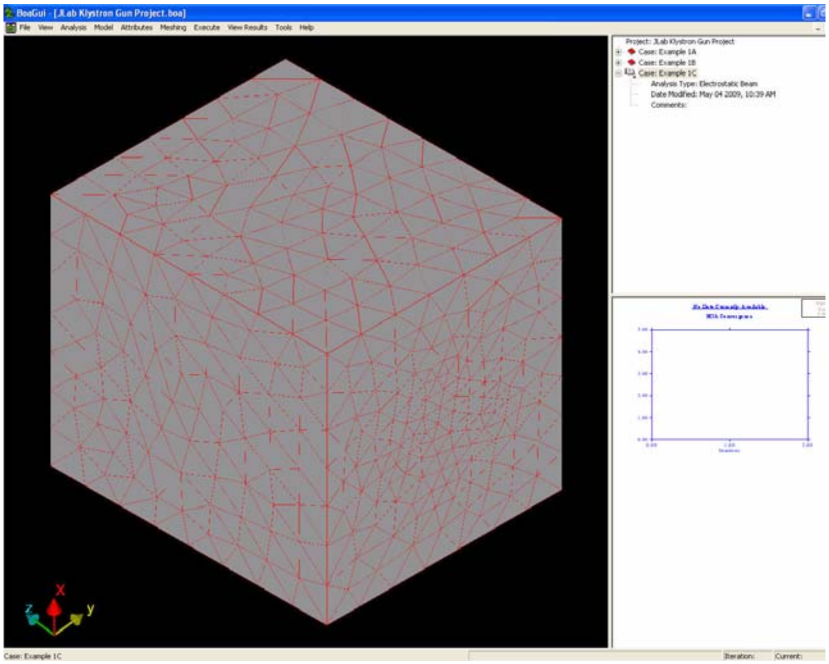
FIGURE 169. The Mod Anode. (Left image is prior to defining the ModAnodeSurface model surface. Right image is after defining the ModAnodeSurface model surface.)

8. Click Enter. The dialog box in Figure 165 will appear. The *Name* box will not be editable. It will be prepopulated with the name ‘ModTunnel’. (This is the name given in the JLAB Gun.SAT file for the named CAD surface consisting of tags 14 and 17.) Set the *Color* to ‘purple’. Check the *Set Mesh Size* checkbox and the *Set Curvature-Based Refinement* checkbox and leave the associated entries at their default values.
9. Click *OK* to complete the surface definition.

10. The newly defined surface will appear in the table in the *Local Mesh Refinement* section, as shown in the third entry of the table in Figure 166.
11. Click *OK* to close the *Electrostatic Mesh Parameters (Optional)* dialog.
12. Click the *Electrostatic* tab. Check the *Visible* checkbox for the Cathode, Focus Electrode and Anode to make them visible again. Uncheck the *Visible* checkbox for the Mod Anode to make it invisible for the remainder of the example. It's not necessary to see it and it hides the interior of the gun.)
13. Open the *Meshing* pulldown menu and select *Generate Initial Electrostatic Mesh.*
14. A DOS window will appear indicating progress during the mesh generation process. When the process completes, the following text will appear at the bottom of the window: “Mesh Generation Complete, mesh file created: JLAB Gun.sms”. !Press any key to close the DOS window and return control back to the user interface.

4.1.3.6 Viewing the Initial Electrostatic Mesh

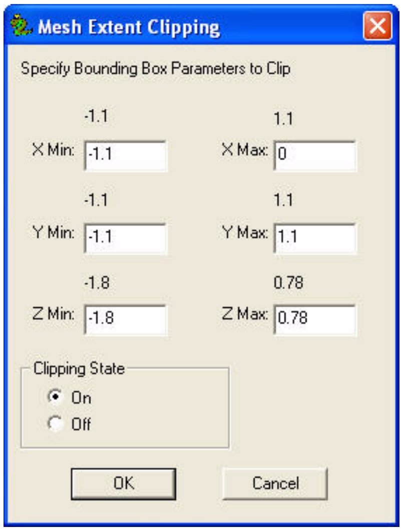
1. To view the initial mesh, open the *Meshing* pulldown menu. Select *Display Initial Mesh* and then select *Complete*. The mesh will appear in the graphics display region.
2. To get the exact orientation shown in Figure 170, click on the *View* pulldown menu and select *Orientation->Isometric*. You may need to use the zoom out feature to view the entire mesh. Right-click on the mouse and drag it backward (or down) until the complete mesh is in view.


BoaGui - [JLab Klystron Gun Project.boa]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
x
y
z
Case: Example 1C
Iteration:
Current:

Project: JLab Klystron Gun Project
Case: Example 1A
Case: Example 1B
Case: Example 1C
Analysis Type: Electrostatic Beam
Date Modified: May 04 2009, 10:39 AM
Comments:

FIGURE 170. Initial Electrostatic Mesh

3. To display only a part of the mesh, use the *Extent Clipping* feature. Open the *Meshing* pull-down menu and select *Extent Clipping Initial Mesh*. The *Mesh Extent Clipping* dialog shown in Figure 171 will appear.



Specify Bounding Box Parameters to Clip

-1.1 1.1
X Min: -1.1 X Max: 0

-1.1 1.1
Y Min: -1.1 Y Max: 1.1

-1.8 0.78
Z Min: -1.8 Z Max: 0.78

Clipping State
● On
○ Off

FIGURE 171. Mesh Extent Clipping dialog

4. Change the *X Max* value to “0” and click *OK*. The partial mesh will appear in the graphics display region as shown in Figure 172.

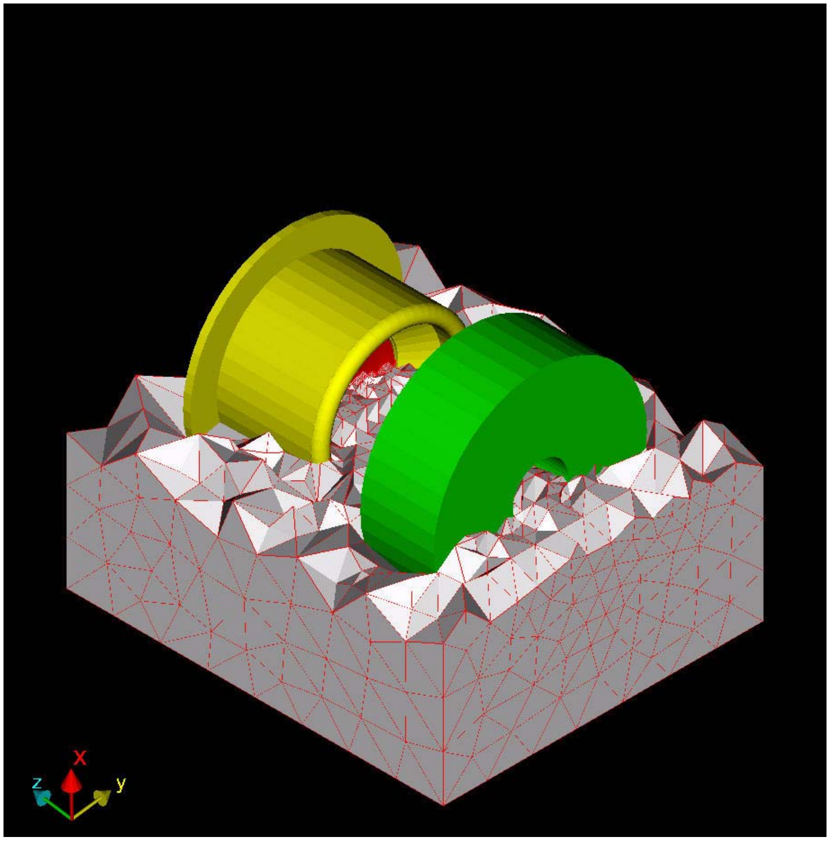
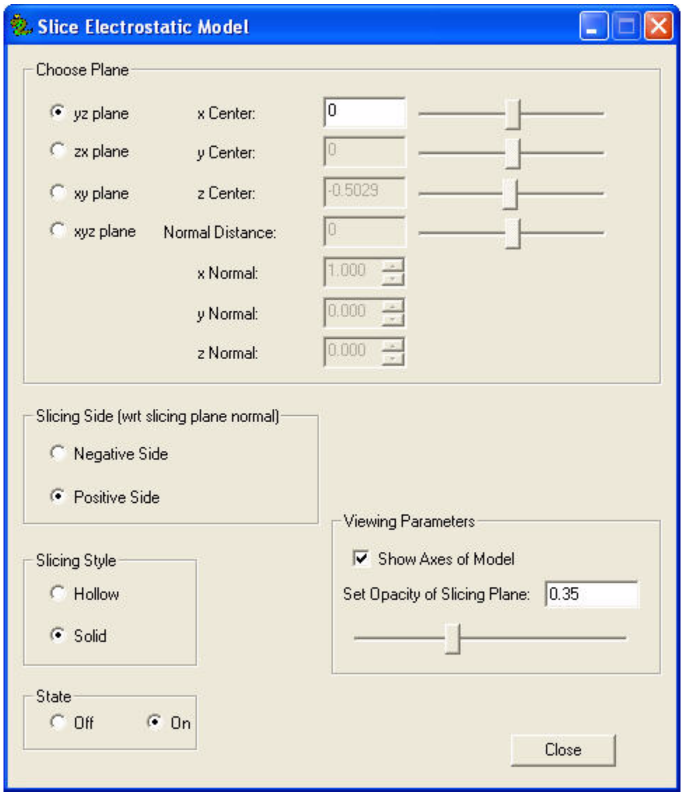


FIGURE 172. Initial Electrostatic Mesh with Extent Clipping

5. To get an even better look at the mesh, slice the model. Click on the *View* menu item and select *Slice Model*. The *Slice Electrostatic Model* dialog shown in Figure 173 will appear. On the *Choose Plane* section, select ‘yz plane’. In the *Slicing Side* section, select ‘Positive Side’. Click



Choose Plane
● yz plane
○ zx plane
○ xy plane
○ xyz plane
x Center: 0
y Center: 0
z Center: -0.5029
Normal Distance: 0
x Normal: 1.000
y Normal: 0.000
z Normal: 0.000

Slicing Side (wrt slicing plane normal)
○ Negative Side
● Positive Side

Slicing Style
○ Hollow
● Solid

State
○ Off ● On

Viewing Parameters
☑ Show Axes of Model
Set Opacity of Slicing Plane: 0.35

FIGURE 173. Slice Electrostatic Model dialog

Close. The image in Figure 174 will appear. Notice that the mesh is much more refined around the thermionic emitter and in the areas you defined as *Local Mesh Refinements* (i.e., ‘Anode-Tunnel, ‘FocusElectrodeSurface’ and ‘ModTunnel’).

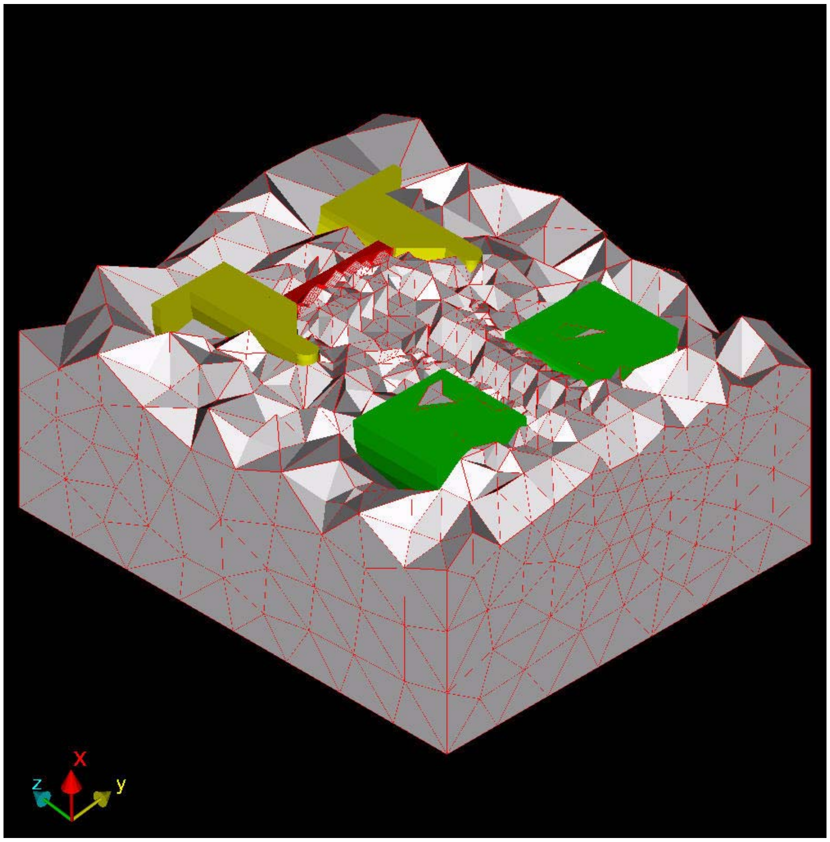


FIGURE 174. Initial Electrostatic Mesh with Extent Clipping and Sliced Model

6. To turn off the *Extent Clipping* feature, reopen the *Mesh Extent Clipping* dialog (i.e., open the *Meshing* pulldown menu and select *Extent Clipping Initial Mesh)* and turn the *Clipping State* radio button to ‘Off’.
7. To hide the initial mesh, open the *Meshing* pulldown. The *Display Initial Mesh->Complete* item is preceded by a check mark, indicating that the initial mesh is displayed. Click it to ‘uncheck’ it and the mesh display will turn off.
8. To show the complete model again, open the *View* menu and select *Slice Model*. On the *Slice Electrostatic Model* dialog, set the *State* radio button to ‘Off’.
9. Click *Close* to close the *Slice Electrostatic Model* dialog.

4.1.3.7 Executing BOA (without Adaptive Meshing)

Once the initial mesh is generated, you can execute the BOA engine.

1. Open the *Execute* pulldown menu and select *Single Case->Execute BOA*.
2. The *Execute Single Case (Electrostatic Analysis)* dialog will appear. (This dialog was shown in Example 1A, Figure 135.)

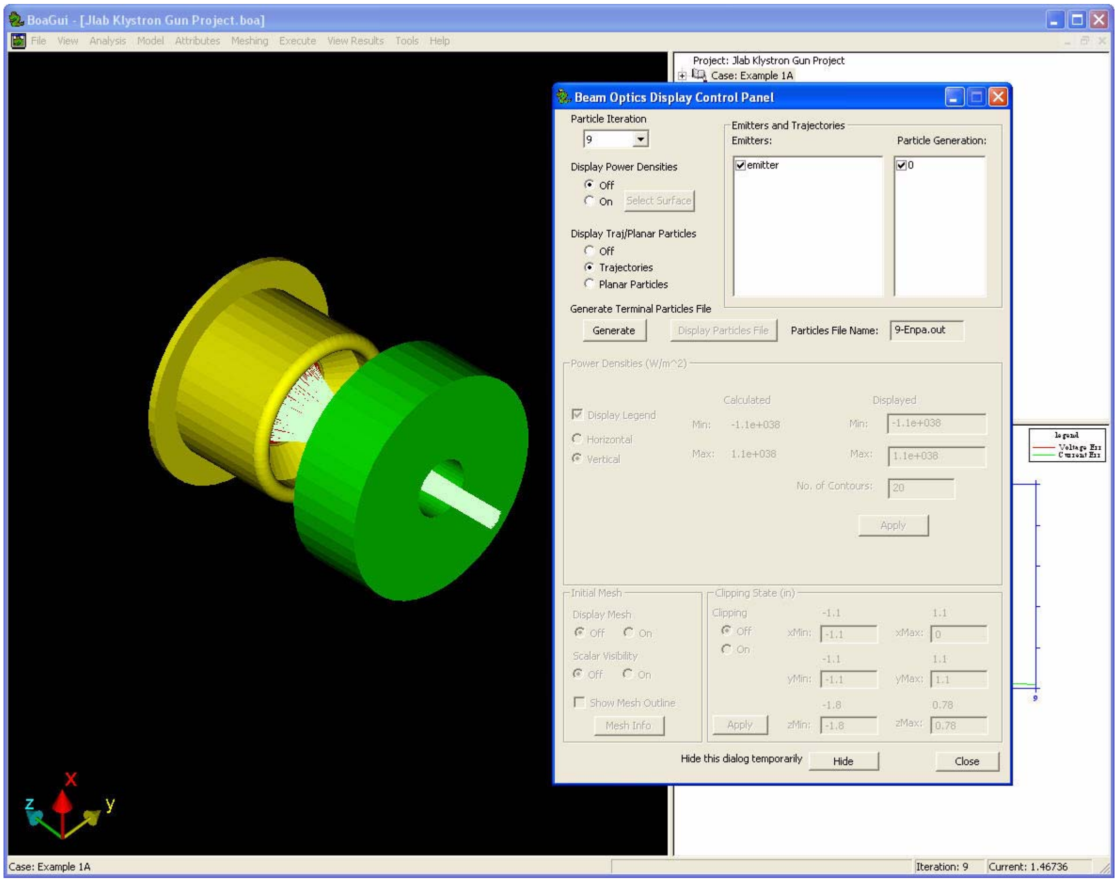
3. Leave all parameters set to the default values. (As in Example 1A, note that *Max Number of Adaptivity Passes* is set to ‘0’. This means the mesh will not be adapted for this run.) Click *Execute*.
4. A DOS window will appear indicating progress during BOA execution. Note that this run will usually take longer than Examples 1A and 1B. When BOA completes, the following text will appear at the bottom of the window “Proceed to View Results but first Press any key to continue/exit”. Hit any key to close the DOS window and return control back to the user interface.

Notice that the lower right corner is used to monitor the progress of the simulation. The convergence plot displays the beam current, the relative errors of the electric field and emission current versus particle iterations. (For a clearer and larger display, you can show the plot in the main window or a new window by going to *View* menu and selecting *Plot Parameters.* On the resulting *Plot Control Parameters dialog*, set the *Plot Type* dropdown box to ‘Convergence’ and the *Plot Display Location* dropdown box to ‘Main Window’ or ‘New Window’.)

4.1.3.8 Viewing Beam Analysis Results

Once BOA completes, you can immediately view the beam optics results.

1. Open the *View Results* pulldown menu. Click on *Beam Analysis*. The *Beam Optics Display Control Panel* dialog will open and the trajectories will automatically appear (per the default settings) as shown in Figure 175.


BoaGui - [Jlab Klystron Gun Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Beam Optics Display Control Panel
Select Surface
Generate
Display Particles File
Apply
Mesh Info
Hide
Close
x
y
z
Legend
Voltage Err
Current Err
9

Project: Jlab Klystron Gun Project
Case: Example 1A

Particle Iteration: 9
Display Power Densities: ● Off ○ On
Display Traj/Planar Particles: ○ Off ● Trajectories ○ Planar Particles
Emitters and Trajectories
Emitters: ☑ emitter
Particle Generation: ☑ 0
Generate Terminal Particles File
Particles File Name: 9-Enpa.out
Power Densities (W/m^2)
☑ Display Legend
○ Horizontal
● Vertical

	Calculated	Displayed
Min:	-1.1e+038	-1.1e+038
Max:	1.1e+038	1.1e+038

No. of Contours: 20

Initial Mesh
Display Mesh: ● Off ○ On
Scalar Visibility: ● Off ○ On
☐ Show Mesh Outline

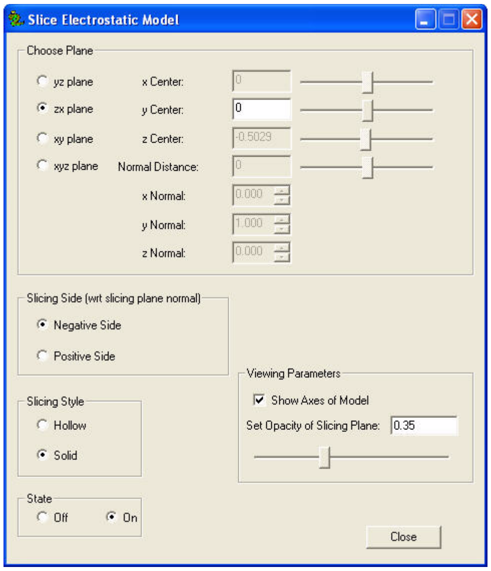
Clipping State (in)
Clipping: ● Off ○ On
xMin: -1.1 (-1.1) xMax: 0 (1.1)
yMin: -1.1 (-1.1) yMax: 1.1 (1.1)
zMin: -1.8 (-1.8) zMax: 0.78 (0.78)

Hide this dialog temporarily

Case: Example 1A
Iteration: 9
Current: 1.46736

FIGURE 175. Trajectories On (shown with the model sliced)

2. To get a better look at the trajectories, slice the model. First click on the *View* menu item and select *Slice Model*. The *Slice Electrostatic Model* dialog will appear (Figure 176).
3. In the *Choose Plane* section, select the ‘zx’ radio button.
4. In the *Slicing Side* section, select ‘Negative Side’ radio button.
5. Click *Close*. The model will be sliced in half, exposing the trajectories, as shown in Figure 177.
6. Use the right-mouse button to zoom in on the trajectories, as desired.
7. To view the intersection of the trajectory particles on a desired plane, change the *Display Traj/Planar Particles* radio button to ‘Planar Particles’.


Slice Electrostatic Model
Close

Choose Plane
○ yz plane
● zx plane
○ xy plane
○ xyz plane

x Center: 0
y Center: 0
z Center: -0.5029
Normal Distance: 0
x Normal: 0.000
y Normal: 1.000
z Normal: 0.000

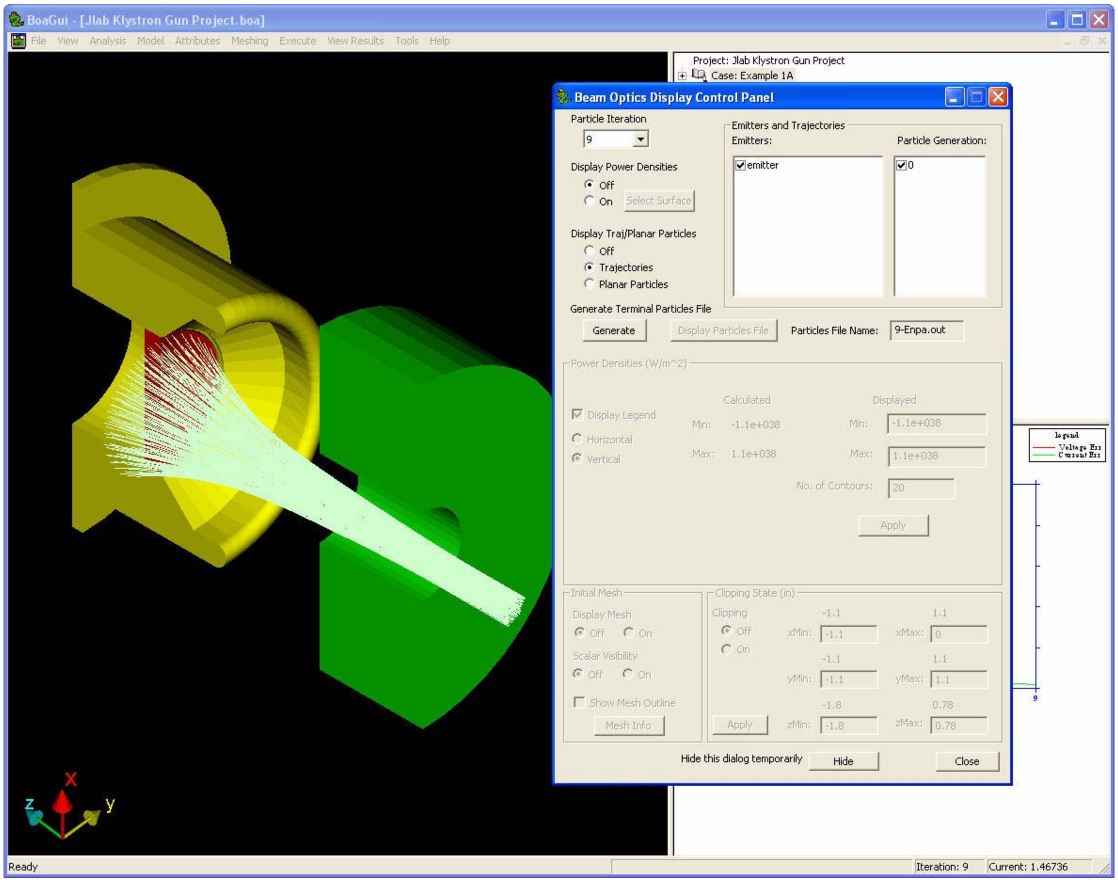
Slicing Side (wrt slicing plane normal)
● Negative Side
○ Positive Side

Slicing Style
○ Hollow
● Solid

State
○ Off ● On

Viewing Parameters
☑ Show Axes of Model
Set Opacity of Slicing Plane: 0.35

FIGURE 176. Slice Electrostatic Model dialog


BoaGui - [Jlab Klystron Gun Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
Beam Optics Display Control Panel
Select Surface
Generate
Display Particles File
Apply
Mesh Info
Hide
Close
x
y
z
Ready

Project: Jlab Klystron Gun Project
Case: Example 1A

Particle Iteration: 9

Display Power Densities
● Off
○ On

Display Traj/Planar Particles
○ Off
● Trajectories
○ Planar Particles

Emitters and Trajectories
Emitters: ☑ emitter
Particle Generation: ☑ 0

Generate Terminal Particles File
Particles File Name: 9-Enpa.out

Power Densities (W/m^2)
☑ Display Legend
○ Horizontal
● Vertical

	Calculated	Displayed
Min:	-1.1e+038	-1.1e+038
Max:	1.1e+038	1.1e+038

No. of Contours: 20

Initial Mesh
Display Mesh ● Off ○ On
Scalar Visibility ● Off ○ On
☐ Show Mesh Outline

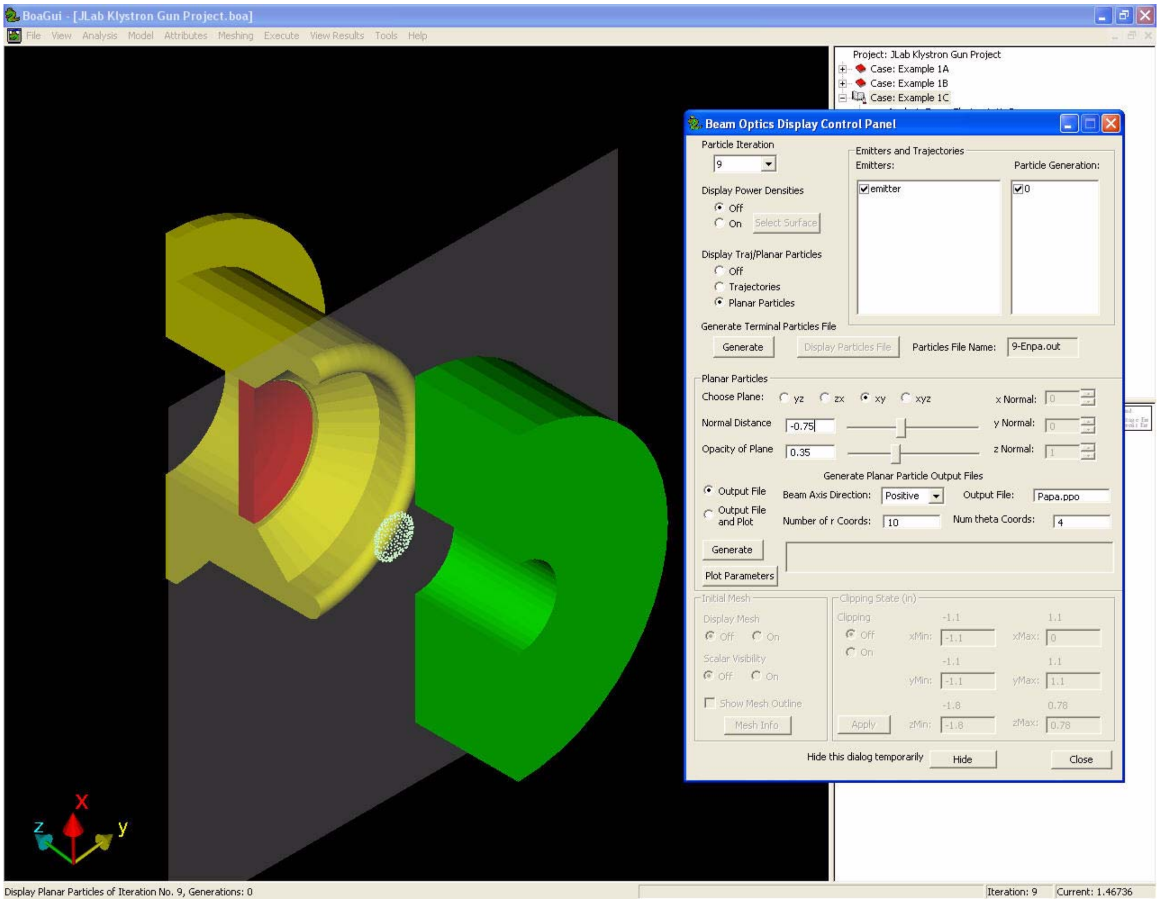
Clipping State (in)
Clipping ● Off ○ On
xMin: -1.1 xMax: 0
yMin: -1.1 yMax: 1.1
zMin: -1.8 zMax: 0.78

Hide this dialog temporarily

Iteration: 9
Current: 1.46736

FIGURE 177. Trajectories On (shown with the model sliced)

8. In the *Planar Particles* section, leave the *Choose Plane* radio button set to 'xy', and use the *Normal Distance* edit box or slider to define a desired distance. In Figure 178, the *Normal Distance* is set to -0.75.


BoaGui - [JLab Klystron Gun Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
x
y
z
Select Surface
Generate
Display Particles File
Generate
Plot Parameters
Mesh Info
Apply
Hide
Close

Project: JLab Klystron Gun Project
Case: Example 1A
Case: Example 1B
Case: Example 1C

Beam Optics Display Control Panel

Particle Iteration: 9

Display Power Densities: ● Off ○ On

Display Traj/Planar Particles: ○ Off ○ Trajectories ● Planar Particles

Emitters and Trajectories
Emitters: ☑ emitter
Particle Generation: ☑ 0

Generate Terminal Particles File
Particles File Name: 9-Enpa.out

Planar Particles
Choose Plane: ○ yz ○ zx ● xy ○ xyz
Normal Distance: -0.75
Opacity of Plane: 0.35
x Normal: 0
y Normal: 0
z Normal: 1

Generate Planar Particle Output Files
● Output File
○ Output File and Plot
Beam Axis Direction: Positive
Output File: Papa.ppo
Number of r Coords: 10
Num theta Coords: 4

Initial Mesh
Display Mesh: ● Off ○ On
Scalar Visibility: ● Off ○ On
☐ Show Mesh Outline

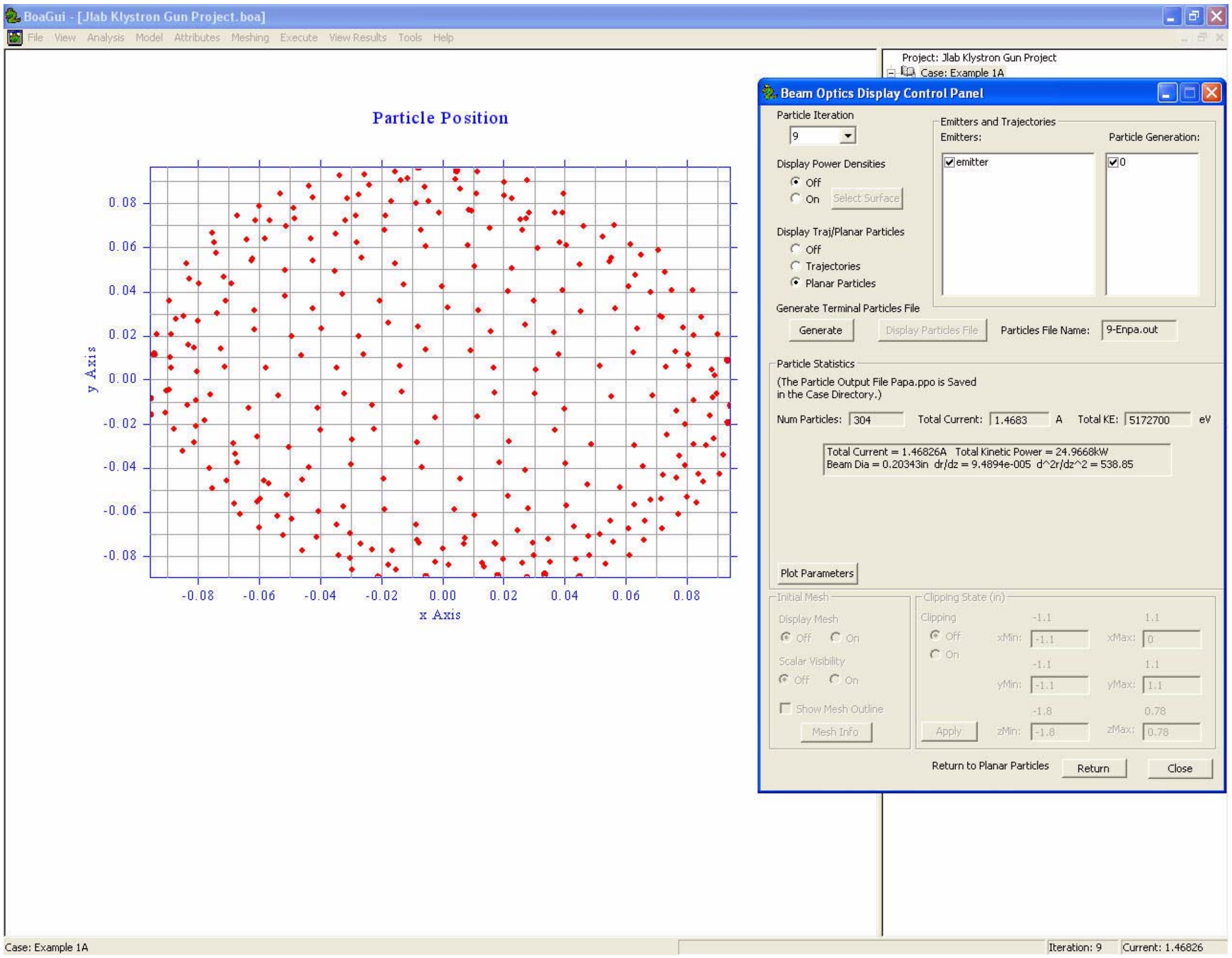
Clipping State (in)
Clipping: ● Off ○ On
xMin: -1.1 (-1.1) xMax: 0 (1.1)
yMin: -1.1 (-1.1) yMax: 1.1 (1.1)
zMin: -1.8 (-1.8) zMax: 0.78 (0.78)

Hide this dialog temporarily

Display Planar Particles of Iteration No. 9, Generations: 0
Iteration: 9
Current: 1.46736

FIGURE 178. Planar Particles

9. Next, generate a coordinated plot of the planar particles and create an output file. Select the *Output File and Plot* radio button and click *Generate*. The coordinated particle plot appears in the main window as shown in Figure 179. The detailed beam profile characteristics appear in the *Plot Statistics* section of the *Beam Optics Display Control Panel* dialog. The particle output file named ‘papa.ppo’ is saved to the case directory.


BoaGui - [Jlab Klystron Gun Project.boa]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
Particle Position
0.08
0.06
0.04
0.02
0.00
-0.02
-0.04
-0.06
-0.08
y Axis
-0.08
-0.06
-0.04
-0.02
0.00
0.02
0.04
0.06
0.08
x Axis
Beam Optics Display Control Panel
Generate
Display Particles File
Select Surface
Plot Parameters
Mesh Info
Apply
Return
Close

Project: Jlab Klystron Gun Project
Case: Example 1A

Particle Iteration: 9

Display Power Densities
● Off
○ On

Display Traj/Planar Particles
○ Off
○ Trajectories
● Planar Particles

Emitters and Trajectories
Emitters: ☑ emitter
Particle Generation: ☑ 0

Generate Terminal Particles File
Particles File Name: 9-Enpa.out

Particle Statistics
(The Particle Output File Papa.ppo is Saved in the Case Directory.)

Num Particles: 304 Total Current: 1.4683 A Total KE: 5172700 eV

Total Current = 1.46826A Total Kinetic Power = 24.9668kW
Beam Dia = 0.20343in dr/dz = 9.4894e-005 d^2r/dz^2 = 538.85

Initial Mesh
Display Mesh: ● Off ○ On
Scalar Visibility: ● Off ○ On
☐ Show Mesh Outline

Clipping State (in)
Clipping: ● Off ○ On
xMin: -1.1 (-1.1) xMax: 0 (1.1)
yMin: -1.1 (-1.1) yMax: 1.1 (1.1)
zMin: -1.8 (-1.8) zMax: 0.78 (0.78)

Return to Planar Particles

Case: Example 1A Iteration: 9 Current: 1.46826

FIGURE 179. Coordinated Plot of planar Particles

10.To return to the pre-plot state of the *Beam Optics Display Control Panel dialog*, click *Return*. Experiment with other parameters and results, as desired. To close the *Beam Optics Display Control Panel*, click *Close*.

If desired, you can also view the Electrostatic results (as you did in Examples 1A and 1B).

Before closing the case, make sure to save it. Right-click on the case name in the Project/Case Tree Control and select *Save Case*.

4.2 Example 2 - Multi-stage Depressed Collector (MSDC)

In this example, BOA is used to perform both an Electrostatic Beam analysis and an Electro/Magnetostatic Beam analysis on a multi-staged depressed collector (MSDC). A cross section of the is shown in Figure 180. An injected beam enters through a opening in the body of the shown on the right side of the diagram. The voltage on both the body and stage 1 is 25000 Volts. The voltage is 16000 volts on stage 2, 8000 Volts on stage 3 and 0 Volts Stage 4.

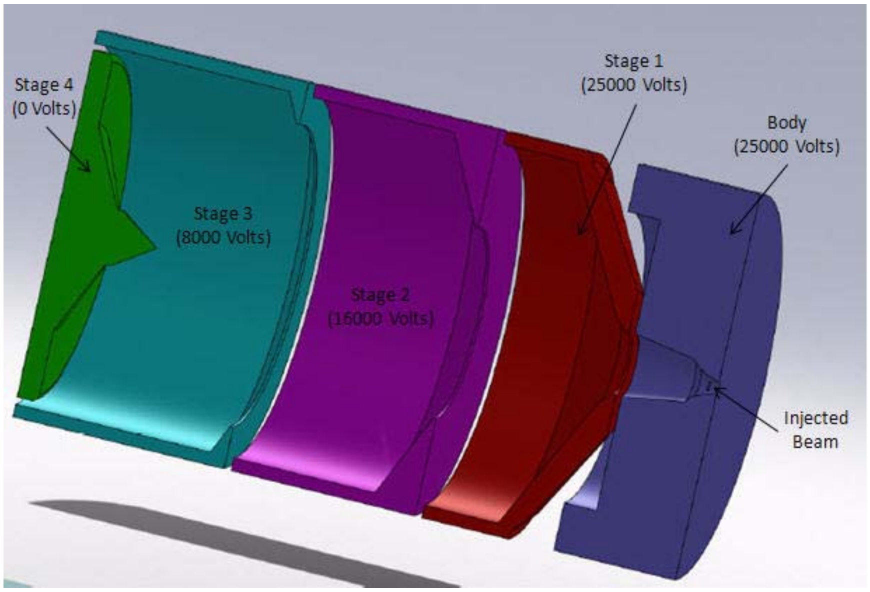


FIGURE 180. Cross section of Multi-stage Depressed Collector

This example consists of two different cases. Example 2A illustrates a basic electrostatic beam analysis with an injected beam. Example 2B builds on 2A but adds an external magnetic field and secondary emitter surfaces.

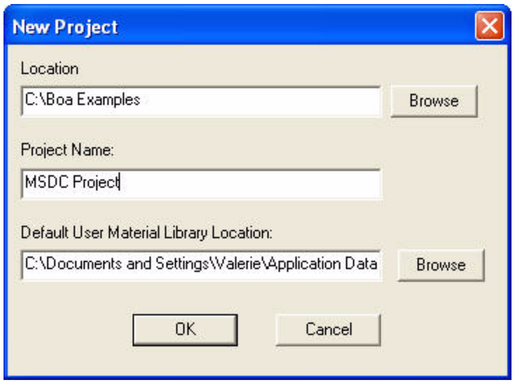
4.2.1 Example 2A - Electrostatic Beam Analysis (with Injected Beam)

4.2.1.1 Getting Started

1. Open BoaGui.exe by clicking on the BoaGui icon on your desktop or the BoaGui icon in the Start menu.

4.2.1.2 Creating a New Project

1. Go to the *File* pulldown menu and select *New Project*. The dialog in Figure 181 will appear. Unlike the screen shown in Figure 181, initially the first two data entry boxes will be blank.
2. Use the *Browse* button to locate the ‘Boa Examples’ directory that you created in Example 1.
3. Specify the *Project Name* in the box provided. In this example, the *Project Name* is ‘ Project’.
4. Leave the *User Material Library Location* as is.
   It is not used in this example. (This directory automatically defaults to your computer’s home directory.)



Location: C:\Boa Examples

Project Name: MSDC Project

Default User Material Library Location: C:\Documents and Settings\Valerie\Application Data

FIGURE 181. New Project Dialog Box

5. Click *OK*. The new Project will appear in the Project/Case Manager tree control on the right side of the BoaGui screen. Behind the scenes, a Project folder will be created in the specified location. The project folder will contain the BOA Project file (*.boa) and all associated Case folders.

4.2.1.3 Creating a New Case

1. Right click on ‘ Project’ in the Project/Case Manager tree control. The popup menu shown in Figure 182 will appear.
2. Select *Add New Case*. The dialog shown in Figure 183 will appear. (Initially the data entry fields will be empty.)

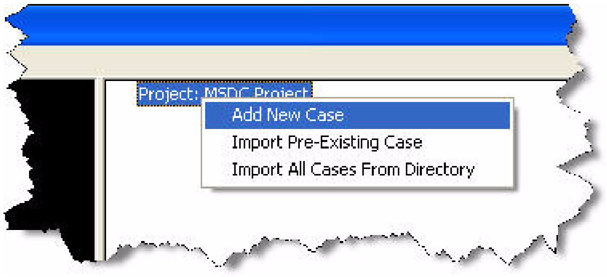
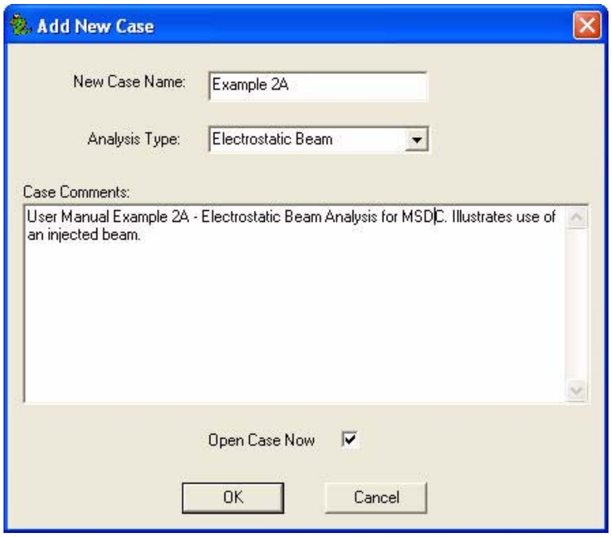


FIGURE 182. The Project/Case Tree Control (shown with right-click menu opened)

3. Specify the *Case Name* in the box provided. In this example the *Case Name* is ‘Example 2A’.
4. Select ‘Electrostatic Beam’ in the *Analysis Type* dropdown box.
5. If desired, add *Case Comments* to describe the case as shown in Figure 183.
6. Leave the *Open Case Now* checkbox checked.
7. Click *OK*. The new Case will appear beneath the Project in the Project/Case Manager tree control as shown in Figure 184. Behind the scenes, a new Case folder will be placed in the current Project folder. The case folder will include all files specific to the case.



New Case Name: Example 2A

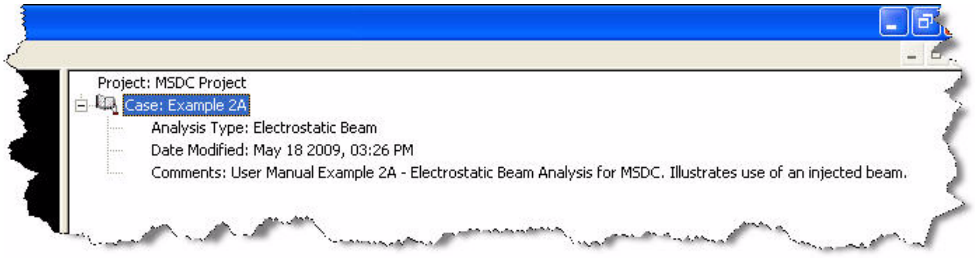
Analysis Type: Electrostatic Beam

Case Comments:

User Manual Example 2A - Electrostatic Beam Analysis for MSDC. Illustrates use of an injected beam.

Open Case Now ☑

FIGURE 183. Add New Case dialog



Project: MSDC Project

Case: Example 2A

Analysis Type: Electrostatic Beam

Date Modified: May 18 2009, 03:26 PM

Comments: User Manual Example 2A - Electrostatic Beam Analysis for MSDC. Illustrates use of an injected beam.

FIGURE 184. Project/Case Tree Control (with Case opened)

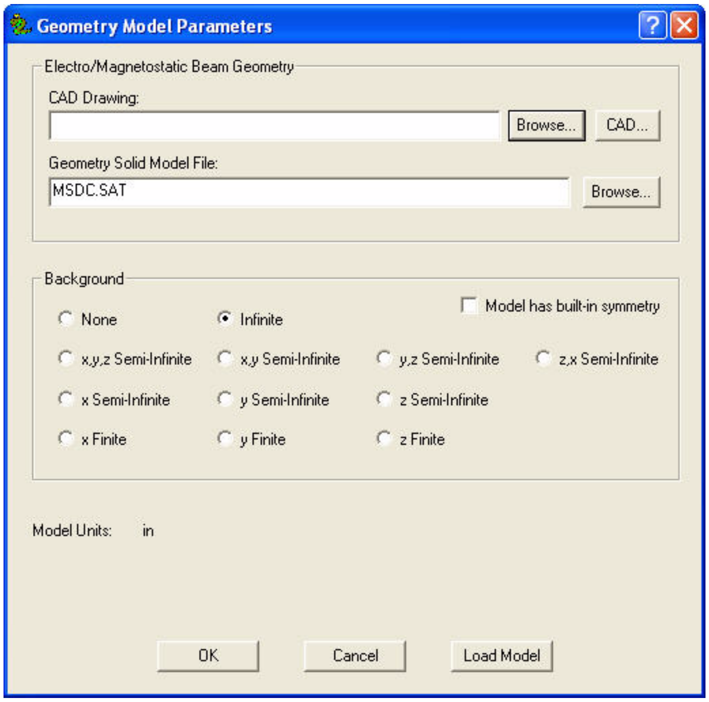
4.2.1.4 Saving a Case

As you continue through this example, it is a good idea to save your work occasionally. To save a case, right click on the *Case* name in the Project/Case menu bar and select *Save Case*. (Alternatively, open the *File* pulldown menu and click *Save Case*.)

4.2.1.5 Specifying the Geometry File

Next, specify the geometry file on which to base the analysis. The Boa installation includes the geometry file you will need to complete this example as discussed below.

1. Click on the *Model* menu item and the dialog shown in Figure 185 will appear. (Unlike Figure 185, initially the *Geometry Solid Model File* data entry box will be blank.)
2. Click the *Browse* button next to the *Geometry Solid Model File* entry box to locate the geometry file. This will bring up a Browse window that allows you to locate the desired geometry file.
3. Locate the file '.SAT'. If you installed BOA using the default directory settings, this file will be located in C:/Program Files/CCR/Beam Optics Analyzer/Examples/CAD Files. Select the file and click *Open* (or alternatively, double-click on the file).


Geometry Model Parameters

Electro/Magnetostatic Beam Geometry

CAD Drawing:

Geometry Solid Model File: MSDC.SAT

Background

○ None ● Infinite ☐ Model has built-in symmetry

○ x,y,z Semi-Infinite ○ x,y Semi-Infinite ○ y,z Semi-Infinite ○ z,x Semi-Infinite

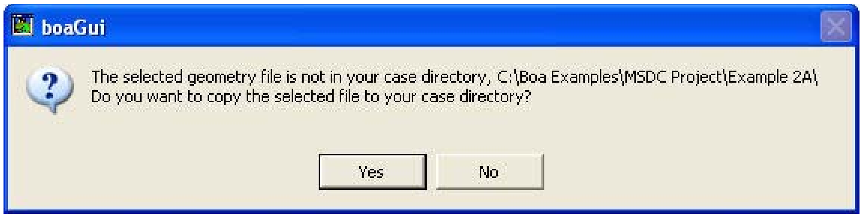
○ x Semi-Infinite ○ y Semi-Infinite ○ z Semi-Infinite

○ x Finite ○ y Finite ○ z Finite

Model Units: in

FIGURE 185. Geometry Model Parameters dialog

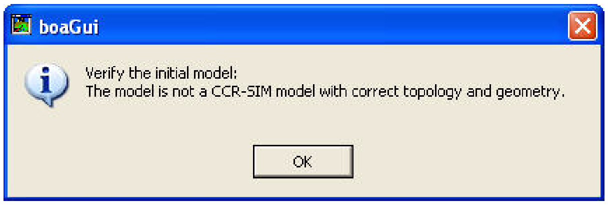
4. The informational message shown in Figure 186 will inform you that BoaGui will create a copy of the file and place it in the current Case directory. Click *Yes*.


boaGui

The selected geometry file is not in your case directory, C:\Boa Examples\MSDC Project\Example 2A\
Do you want to copy the selected file to your case directory?

FIGURE 186. Geometry File Copy Message

5. The name of the geometry file will appear in the *Geometry Solid Model File* entry box, as shown in Figure 185.

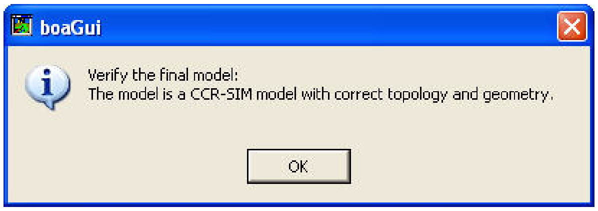

boaGui

Verify the initial model:
The model is not a CCR-SIM model with correct topology and geometry.

FIGURE 187. Initial Model Verification Message

6. The *Model Units* will appear undefined until you load the model. Click *Load Model* and an informational message will inform you that the initial model has been loaded and verified, as shown in Figure 187. Click *OK*.

7. A second informational message will inform you that the final model has been loaded and verified as shown in Figure 188. Click *OK*. (The initial model is the model you specified, as is. The final model is the initial model plus modifications necessary to make it usable by BOA.)



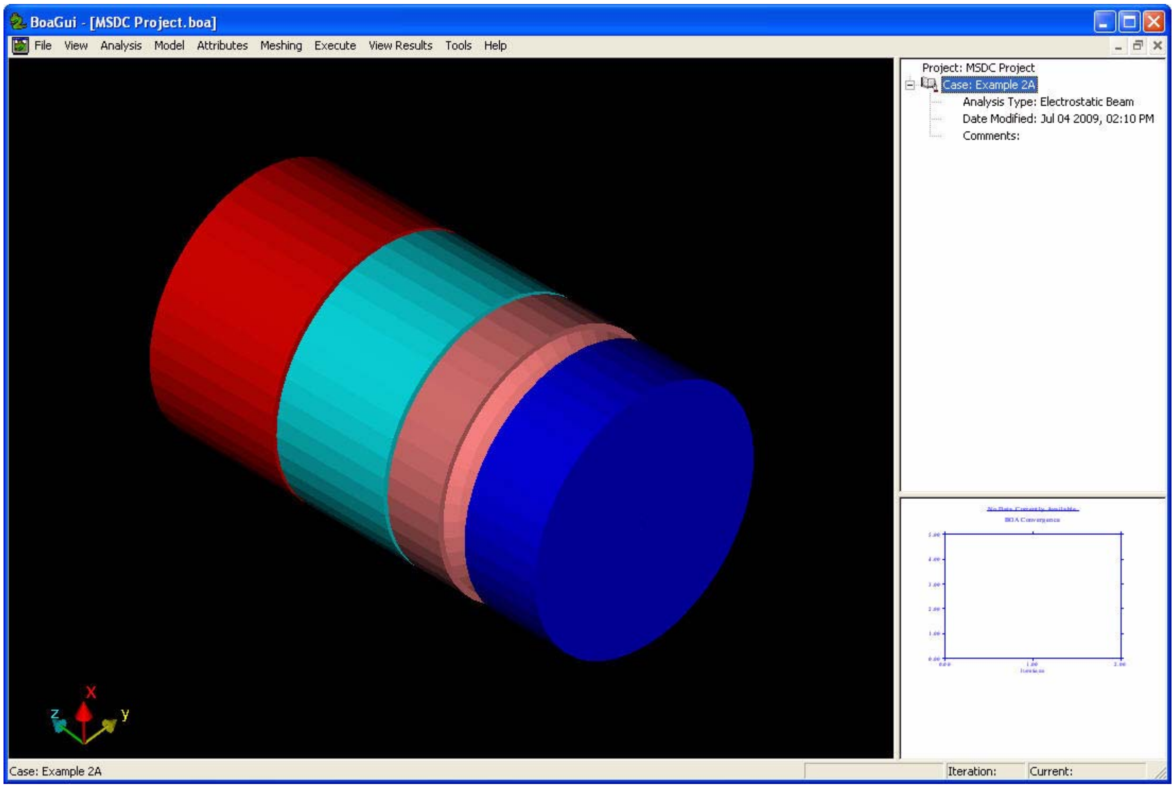
Verify the final model:
The model is a CCR-SIM model with correct topology and geometry.

FIGURE 188. Final Model Verification Message

8. The model units used in the SAT file will appear at the bottom of the *Geometry Model Parameters* dialog. In this case, the units are set to ‘inches’.

9. Click *OK* to exit the *Geometry Model Parameters* dialog.

10. The geometry model will appear in the graphics display region as shown in Figure 189.


BoaGui - [MSDC Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
x
z
y
No Data Currently Available
BOA Convergence
Iteration
Iteration:
Current:

Project: MSDC Project
Case: Example 2A
Analysis Type: Electrostatic Beam
Date Modified: Jul 04 2009, 02:10 PM
Comments:

Case: Example 2A

FIGURE 189. Geometry Model for MSDC

4.2.1.6 Viewing the Geometry Model

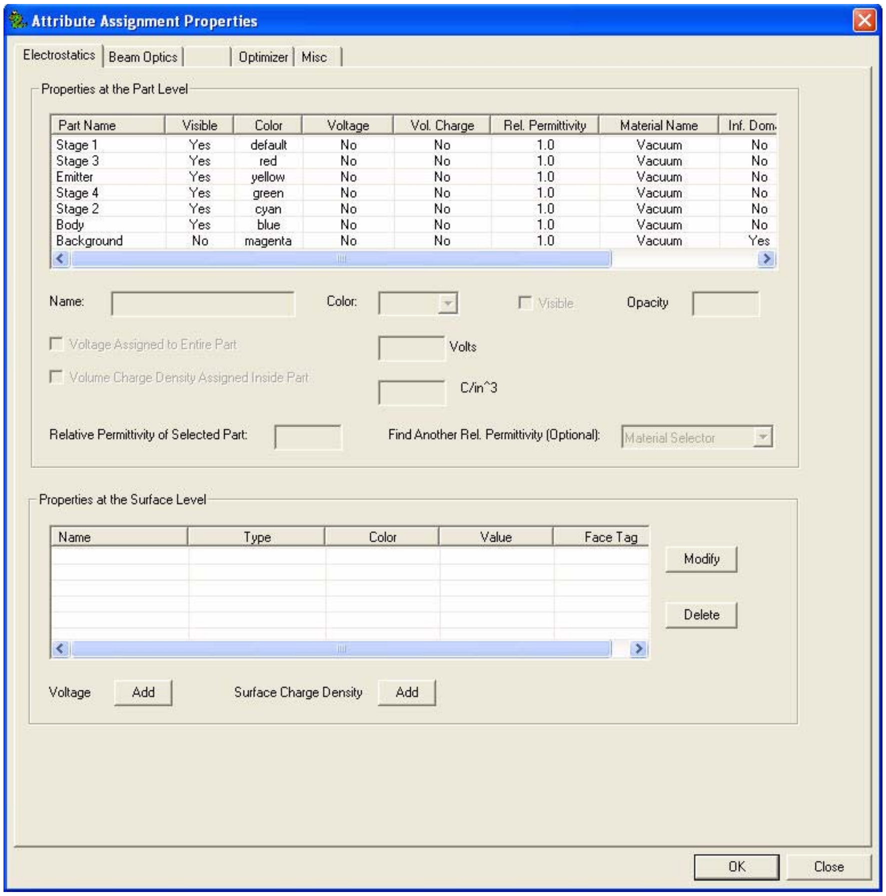
1. Experiment with moving the model around in the graphics display window. To spin the model around, hold down the left mouse button and move the mouse around. To zoom in, hold down the right mouse button and move the mouse forward (or ‘up’). To zoom out, hold down the right mouse button and move the mouse backward (or ‘down’). (Note: These directions assume

you are in the *Trackball* mode, which is the default. If you are in *Joystick* mode, see Section 3.5 for details on mouse control.) You may also want to experiment with different settings on the *View* menu, such as predefined *Orientations*, and different options for *Surface Representation* (‘Wireframe’ vs. ‘Surface’) and *View Representation* (‘Orthogonal’ vs. ‘Perspective’).

2. To return to the orientation shown in Figure 129, click *View->Reset View.*

4.2.1.7 Assigning Electrostatic Attributes

Now you are ready to begin assigning attributes.



Attribute Assignment Properties
Electrostatics
Beam Optics
Optimizer
Misc
Properties at the Part Level
Name:
Color:
☐ Visible
Opacity
☐ Voltage Assigned to Entire Part
Volts
☐ Volume Charge Density Assigned Inside Part
C/in^3
Relative Permittivity of Selected Part:
Find Another Rel. Permittivity (Optional):
Material Selector
Properties at the Surface Level
Modify
Delete
Voltage
Add
Surface Charge Density
Add
OK
Close

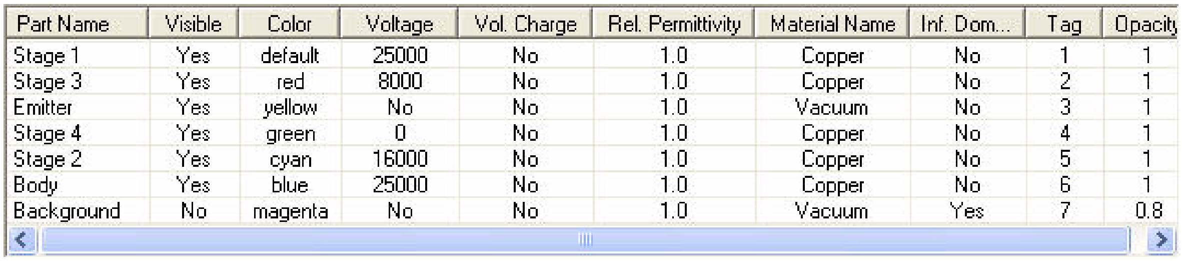
Part Name	Visible	Color	Voltage	Vol. Charge	Rel. Permittivity	Material Name	Inf. Dom.
Stage 1	Yes	default	No	No	1.0	Vacuum	No
Stage 3	Yes	red	No	No	1.0	Vacuum	No
Emitter	Yes	yellow	No	No	1.0	Vacuum	No
Stage 4	Yes	green	No	No	1.0	Vacuum	No
Stage 2	Yes	cyan	No	No	1.0	Vacuum	No
Body	Yes	blue	No	No	1.0	Vacuum	No
Background	No	magenta	No	No	1.0	Vacuum	Yes

Name	Type	Color	Value	Face Tag

FIGURE 190. Attribute Assignment dialog box, Electrostatic tab

1. Click on the *Attributes* menu item. The *Attribute Assignment Properties* dialog shown in Figure 190 will appear, with the *Electrostatics* tab active.
2. The table in the *Properties at the Part Level* section shows all parts contained in the geometry file. The part names are identical to the names provided in the geometry file and they are not editable. Initially BoaGui will assign default attributes to all parts. Typically, the user will want to change most or all of the assigned attributes. The table also contains the ‘Background’ part. This part is not in the geometry file. It is created by BoaGui based on the settings in the *Model Geometry Parameters* dialog. The *Visible* checkbox is checked for all parts except the Background part.
3. Highlight the Stage 1 part. Leave the *Color* dropdown box set to ‘default’. Check the *Voltage Assigned to Entire Part* checkbox and enter ‘25000’ in the data entry box. Set the *Find Another Rel. Permittivity (Optional)* dropdown box to ‘Copper’.

4. Highlight the Stage 3 part and assign the attributes shown in Table 2 below.



Part Name	Visible	Color	Voltage	Vol. Charge	Rel. Permittivity	Material Name	Inf. Dom...	Tag	Opacity
Stage 1	Yes	default	25000	No	1.0	Copper	No	1	1
Stage 3	Yes	red	8000	No	1.0	Copper	No	2	1
Emitter	Yes	yellow	No	No	1.0	Vacuum	No	3	1
Stage 4	Yes	green	0	No	1.0	Copper	No	4	1
Stage 2	Yes	cyan	16000	No	1.0	Copper	No	5	1
Body	Yes	blue	25000	No	1.0	Copper	No	6	1
Background	No	magenta	No	No	1.0	Vacuum	Yes	7	0.8

TABLE 2. Attribute Assignments for case

5. Highlight the Emitter part and assign the attributes shown in Table 2 above.
6. Highlight the Stage 4 part and assign the attributes shown in Table 2 above.
7. Highlight the Stage 2 part and assign the attributes shown in Table 2 above.
8. Highlight the Body part and assign the attributes shown in Table 2 above.
9. You have now completed the definition of properties at the part level. (The *Properties at the Surface Level* section allows you to define properties for specific surfaces on parts in the electrostatic geometry model. For this example, there will be no entries in this table.)

4.2.1.8 Assigning Beam Optics Attributes (Injected Emitter)

Next you will assign the beam optics attributes. For this case, this means defining the injected emitter surface and specifying the emitter part as optically transparent.

1. The *Attribute Assignment Properties* dialog should still be open with the *Electrostatic* tab active. In the next few steps, the goal will be to define the surface of the part named ‘Emitter’ as an Injected Emitter. In order to better see the emitter, make Stage 1, Stage 2, Stage 3 and Stage 4 invisible by unchecking the *Visible* checkbox for each.
2. Rotate the model so that the emitter surface is clearly visible as shown in Figure 191. To center the visible model, open the *View* menu and click *Refresh View.*

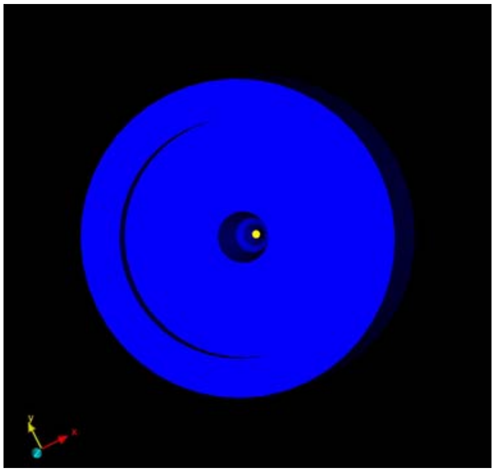
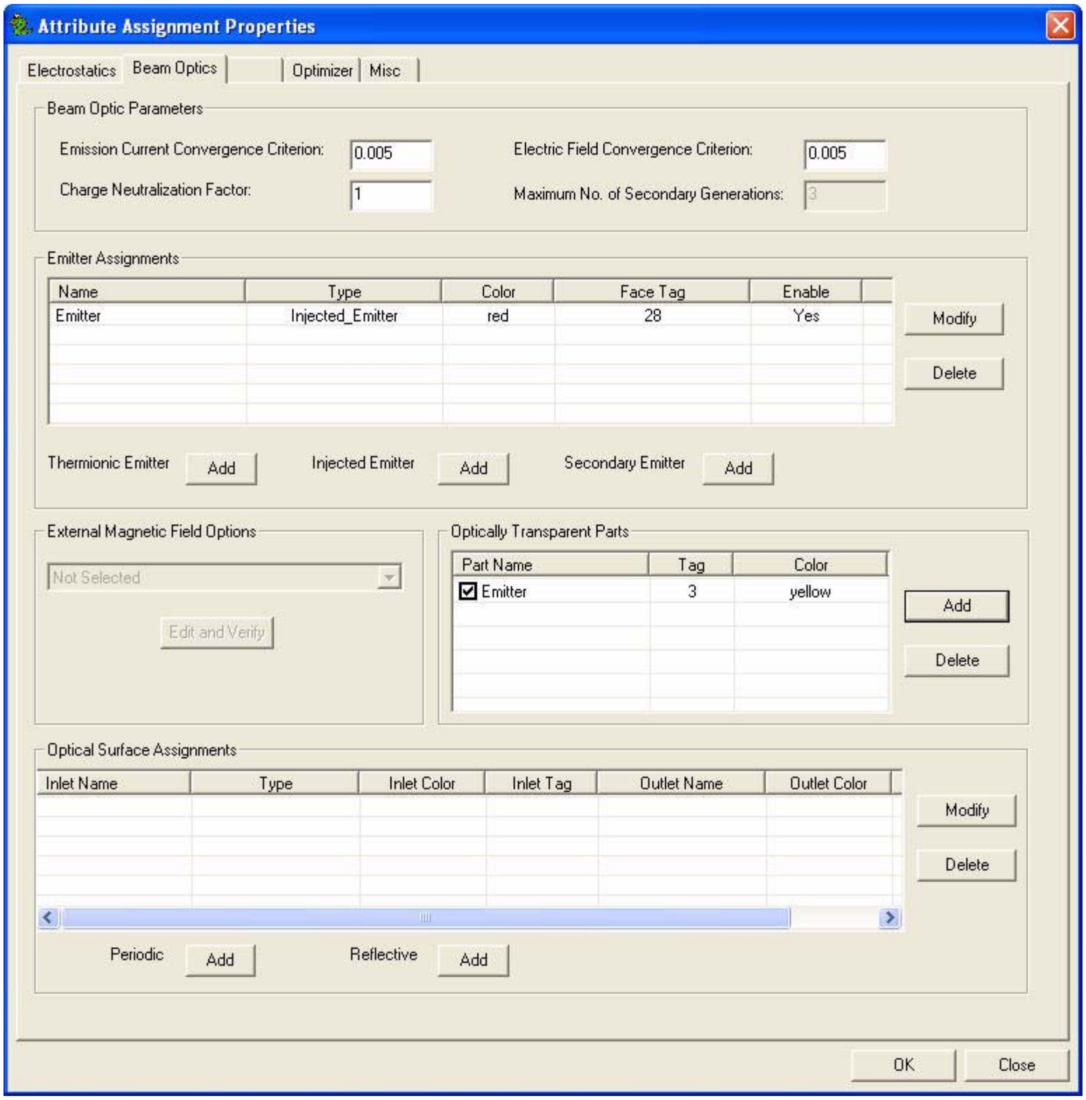


FIGURE 191. Geometry rotated so that emitter surface is visible (in yellow).

3. Click on the *Beam Optics* tab. The dialog shown in Figure 192 will appear. Unlike Figure 192, however, both the *Emitter Assignments* table and the *Optically Transparent Parts* table will be blank.


Attribute Assignment Properties
Electrostatics
Beam Optics
Optimizer
Misc
Beam Optic Parameters
Emitter Assignments
Modify
Delete
Thermionic Emitter
Add
Injected Emitter
Add
Secondary Emitter
Add
External Magnetic Field Options
Not Selected
Edit and Verify
Optically Transparent Parts
Add
Delete
Optical Surface Assignments
Modify
Delete
Periodic
Add
Reflective
Add
OK
Close

Emission Current Convergence Criterion: 0.005

Electric Field Convergence Criterion: 0.005

Charge Neutralization Factor: 1

Maximum No. of Secondary Generations: 3

Name	Type	Color	Face Tag	Enable
Emitter	Injected_Emitter	red	28	Yes

Part Name	Tag	Color
☑ Emitter	3	yellow

Inlet Name	Type	Inlet Color	Inlet Tag	Outlet Name	Outlet Color

FIGURE 192. Attribute Assignment dialog box, Beam Optics tab

4. Leave the *Beam Optic Parameters* set to the defaults.

5. In the *Emitter Assignments* section, click the *Injected Emitter Add* button beneath the table. This will bring up the graphics display window containing the geometry model.

6. Zoom in on the emitter by right-clicking the mouse and moving it forward (or up). Select the emitter surface by pressing Ctrl-Left mouse button. The emitter surface will appear selected (it will change from yellow to green) as shown in Figure 193. Click enter to complete the selection.

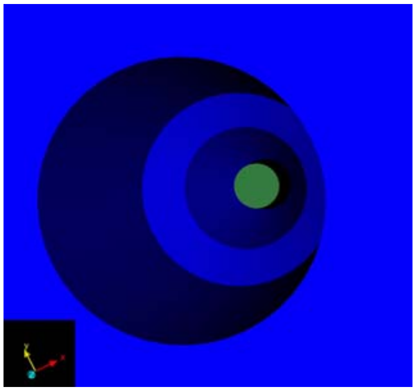
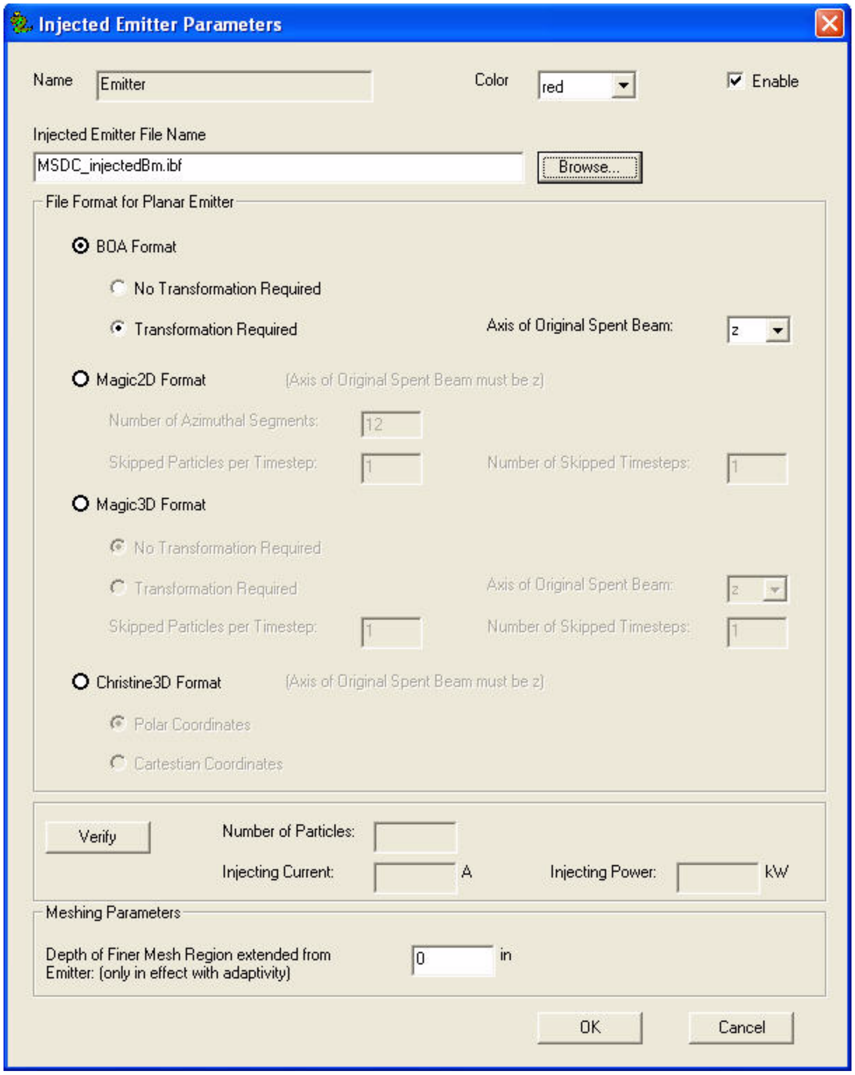


FIGURE 193. Defining the injected emitter

7. The *Injected Emitter Parameters* dialog shown in Figure 194 will appear (though the *Injected Emitter File Name* data entry field will be blank initially). In this case, the surface you selected corresponds to a part surface named ‘Emitter’ in the geometry file. Therefore the *Name* box automatically shows the name ‘Emitter’ and it is not editable. (If the selected surface did not correspond to a named part surface in the geometry model, this field would be editable.) Leave the *Color* dropdown box set to ‘red’.

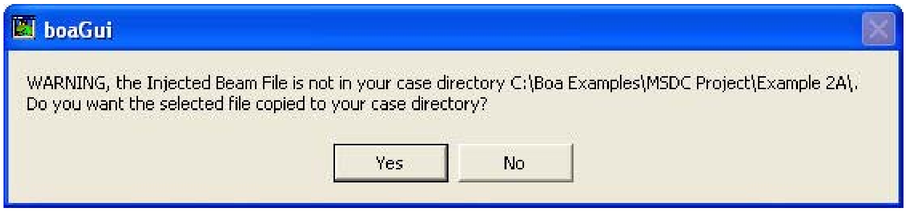


Name: Emitter
Color: red
☑ Enable
Injected Emitter File Name: MSDC_injectedBm.ibf
File Format for Planar Emitter
● BOA Format
○ No Transformation Required
● Transformation Required
Axis of Original Spent Beam: z
○ Magic2D Format (Axis of Original Spent Beam must be z)
Number of Azimuthal Segments: 12
Skipped Particles per Timestep: 1
Number of Skipped Timesteps: 1
○ Magic3D Format
● No Transformation Required
○ Transformation Required
Axis of Original Spent Beam: z
Skipped Particles per Timestep: 1
Number of Skipped Timesteps: 1
○ Christine3D Format (Axis of Original Spent Beam must be z)
● Polar Coordinates
○ Cartesian Coordinates
Number of Particles:
Injecting Current: A
Injecting Power: kW
Meshing Parameters
Depth of Finer Mesh Region extended from Emitter: (only in effect with adaptivity) 0 in

FIGURE 194. Injected Emitter Parameters dialog

8. Use the *Browse* button to locate the injected beam file ‘_InjectedBeam.ibf’. If you installed BOA using the default directory settings, this file will be located in C:/Program Files/CCR/Beam Optics Analyzer/Examples/Data Files. Select the file and click *Open* (or alternatively, double-click on the file). (Note: This beam profile was obtained from running a separate computer code called ‘MAGIC’. You can also create a beam profile using BoaGui’s built-in Cibo tool (*Tools->Cibo)*.)

9. The informational message shown in Figure 195 will inform you that BoaGui will create a copy of the file and place it in the current Case directory. Click *Yes*.



WARNING, the Injected Beam File is not in your case directory C:\Boa Examples\MSDC Project\Example 2A\.
Do you want the selected file copied to your case directory?

FIGURE 195. Injected Beam File Copy Message

10. Back on the *Injected Emitter Parameters* dialog, leave the *File Format for Planar Emitter* radio button set to ‘BOA Format’ with ‘Transformation required’ and *Axis of Original Spent Beam* set to ‘z’.
11. Click *OK* to complete the emitter definition and close the *Injected Emitter Parameters* dialog. The newly defined emitter will appear in the *Emitter Assignments* table as shown in Figure 192.
12. In the *Optically Transparent Parts* section, click the *Add* button next to the table. This will bring up the graphics display window containing the geometry model.
13. Select the Emitter part by pressing Ctrl-Left mouse button anywhere on the part. The entire Emitter part will appear selected (it will change color). Click *Enter* to complete the selection.
14. The part will appear in the *Optically Transparent Parts* table as shown in Figure 192. (This part is considered optically transparent because it is not a physical entity. It models an injected beam.)
15. Click the *Electrostatic* tab. Check the *Visible* checkbox for all parts (except the background) to make them visible again.
16. Click *OK* to close the *Attribute Assignment Properties* dialog.
17. To reset the model position, open the *View* menu and select *Reset View.*

Note: It is a good idea to save your work occasionally. Take this opportunity to save changes to the case by right-clicking on the case name in the Project/Case menu bar and selecting *Save Case*.

4.2.1.9 Generating the Initial Electrostatic Mesh

Once you complete the attribute assignments, you are ready to generate the initial electrostatic mesh. In this example, the default mesh parameters are sufficient so you do not need to open the *Mesh Parameters* dialog.

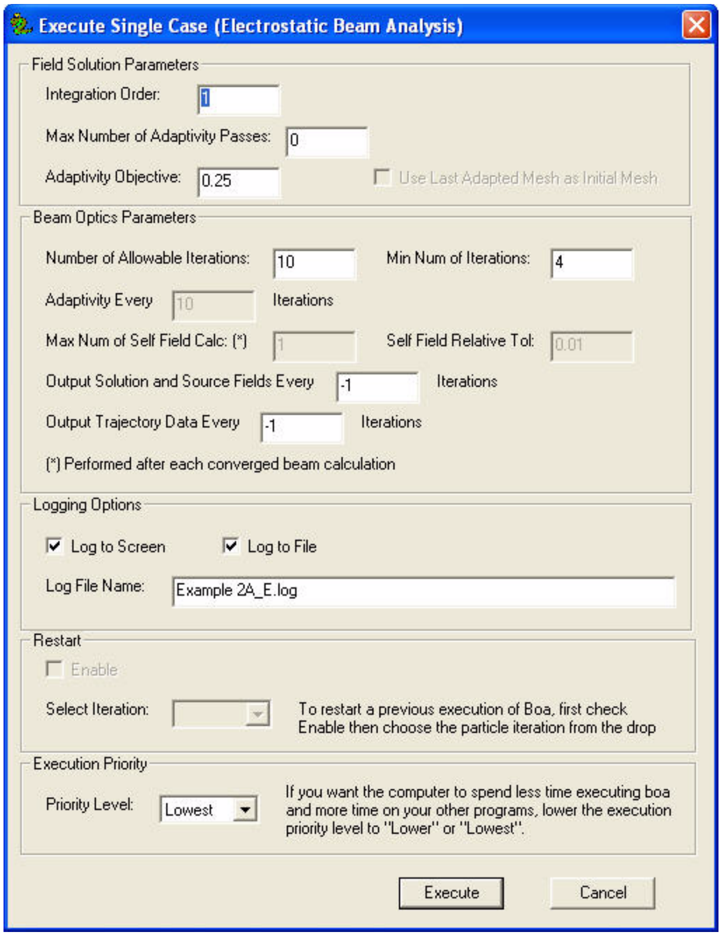
1. Open the *Meshing* pulldown menu and select *Generate Initial Mesh*.
2. A DOS window will appear indicating progress during the mesh generation process. When the process completes, the following text will appear at the bottom of the window: “Mesh Generation Complete, mesh file created: _E.sms !Press any key to continue/exit...”. Hit any key to close the DOS window and return control back to the user interface.

(If desired, you can view the mesh by opening the *Meshing* pulldown menu and selecting *Display Initial Mesh*. The mesh will appear in the graphics display region. This step was illustrated in the previous JLAB example. See Example 1A for details.)

4.2.1.10 Executing BOA

Once the initial mesh is generated, you can execute the BOA engine.

1. Open the *Execute* pulldown menu and select *Single Case->Execute BOA*.
2. The dialog box shown in Figure 196 will appear.
3. Leave all parameters set to the default values. (Note that *Max Number of Adaptivity Passes* is set to ‘0’which means the mesh will not be adapted for this run.) Click *Execute*.
4. A DOS window will appear indicating progress during BOA execution. When BOA completes, the following text will appear at the bottom of the window “Proceed to View Results. Press any key to continue/exit”. Hit any key to close the DOS window and return control back to the user interface.



Execute Single Case (Electrostatic Beam Analysis)

Field Solution Parameters

Integration Order: 1

Max Number of Adaptivity Passes: 0

Adaptivity Objective: 0.25

☐ Use Last Adapted Mesh as Initial Mesh

Beam Optics Parameters

Number of Allowable Iterations: 10

Min Num of Iterations: 4

Adaptivity Every 10 Iterations

Max Num of Self Field Calc: (*) 1

Self Field Relative Tol: 0.01

Output Solution and Source Fields Every -1 Iterations

Output Trajectory Data Every -1 Iterations

(*) Performed after each converged beam calculation

Logging Options

☑ Log to Screen ☑ Log to File

Log File Name: Example 2A_E.log

Restart

☐ Enable

Select Iteration:

To restart a previous execution of Boa, first check Enable then choose the particle iteration from the drop

Execution Priority

Priority Level: Lowest

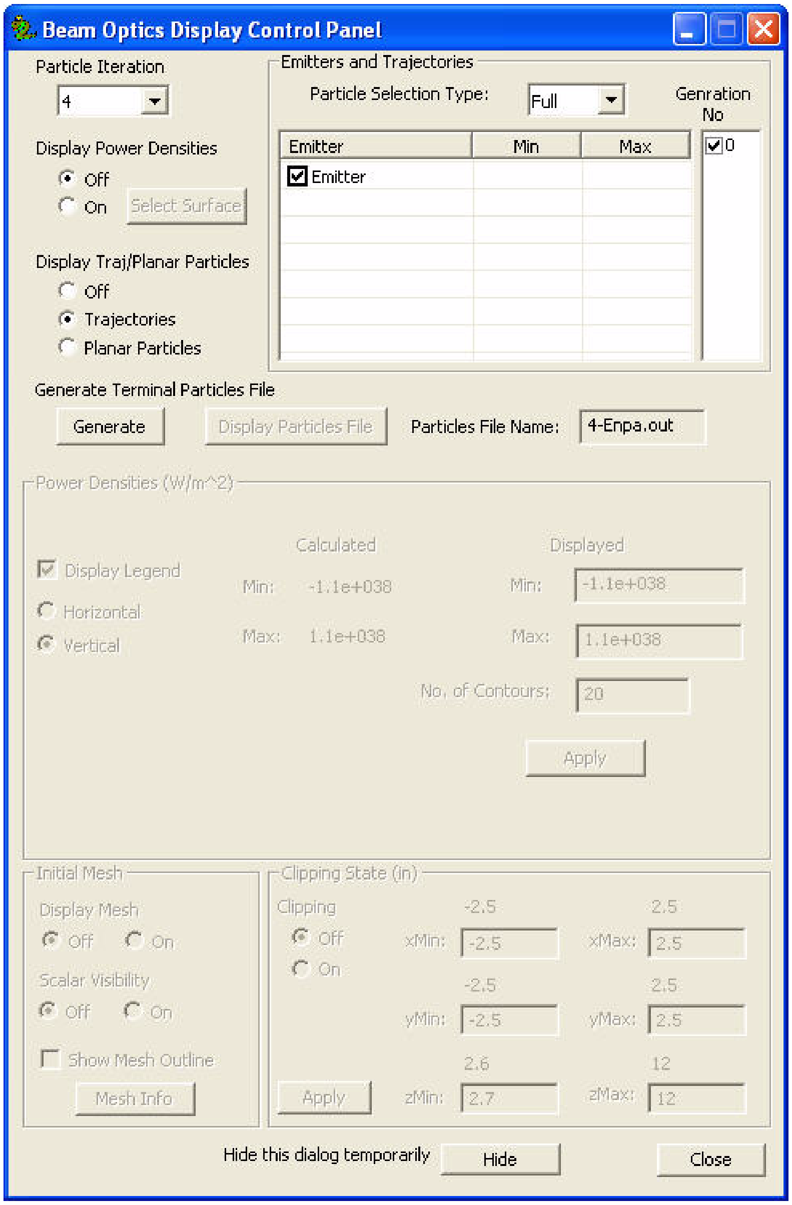
If you want the computer to spend less time executing boa and more time on your other programs, lower the execution priority level to "Lower" or "Lowest".

FIGURE 196. Execute Boa

4.2.1.11 Viewing Beam Analysis Results

For this example, you will proceed directly to the *Beam Analysis* results to view the trajectories and the power densities on the interior surfaces of the collector.

1. Open the *View Results* pulldown menu and select *Beam Analysis*. The *Beam Optics Display Control Panel* dialog shown in Figure 197 will appear.



Beam Optics Display Control Panel

Particle Iteration: 4

Display Power Densities
● Off
○ On

Display Traj/Planar Particles
○ Off
● Trajectories
○ Planar Particles

Emitters and Trajectories
Particle Selection Type: Full
Genration No: ☑ 0

Emitter	Min	Max
☑ Emitter		

Generate Terminal Particles File
Particles File Name: 4-Enpa.out

Power Densities (W/m^2)
☑ Display Legend
○ Horizontal
● Vertical

	Calculated	Displayed
Min:	-1.1e+038	-1.1e+038
Max:	1.1e+038	1.1e+038

No. of Contours: 20

Initial Mesh
Display Mesh: ● Off ○ On
Scalar Visibility: ● Off ○ On
☐ Show Mesh Outline

Clipping State (in)
Clipping: ● Off ○ On

	Min		Max
xMin:	-2.5	xMax:	2.5
yMin:	-2.5	yMax:	2.5
zMin:	2.7	zMax:	12

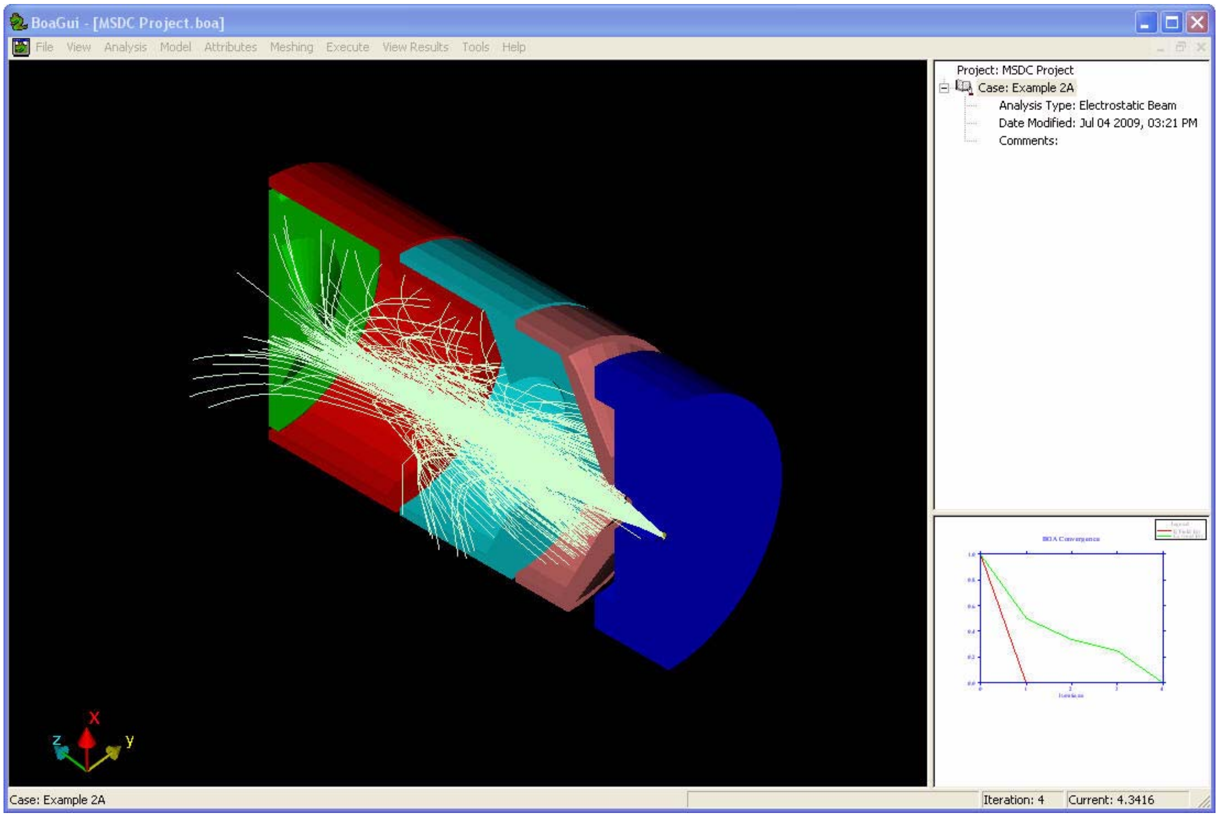
Hide this dialog temporarily

FIGURE 197. Beam Optics Display Control Panel dialog

2. By default, the *Display Traj/Planar Particles* radio button is set to display ‘Trajectories’. In order to view the trajectories inside the collector, you will need to slice the model. Go to the *View* menu and select *Slice Model*.

3. On the resulting *Slice Electrostatic Model* dialog, select the ‘zx plane’ (in the *Choose Plane* section). The trajectories will be exposed as shown in Figure 198. Click *Close* to close the *Slice Electrostatic Model* dialog.

4. Use the mouse to zoom in on the trajectories as desired. When you are done, use the *Reset View* (on the *View* menu) to reset the view.


BoaGui - [MSDC Project.boa]
File View Analysis Model Attributes Meshing Execute View Results Tools Help
BOA Convergence
x
y
z

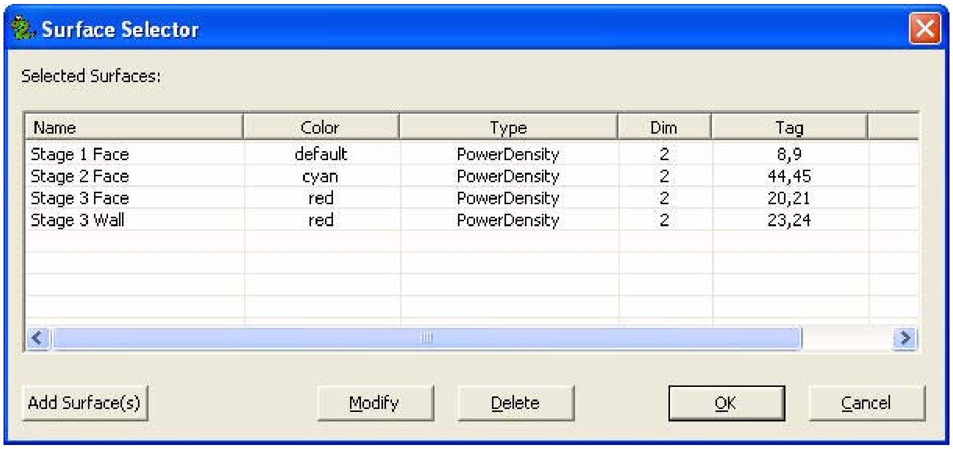
Project: MSDC Project
Case: Example 2A
Analysis Type: Electrostatic Beam
Date Modified: Jul 04 2009, 03:21 PM
Comments:

Case: Example 2A
Iteration: 4
Current: 4.3416

FIGURE 198. Trajectories

5. Next, take a look at the power densities on various surfaces inside the collector. For better viewing, first turn off the trajectories by setting the *Display Traj/Planar Particles* radio button to ‘Off’ on the *Beam Optics Display Control Panel*.

6. Set the *Display Power Densities* radio button to ‘On’. Because you have not selected any surfaces yet, a blank version of the *Surface Selector* dialog shown in Figure 199 will automatically pop up.


Surface Selector
Add Surface(s)
Modify
Delete
OK
Cancel

Selected Surfaces:

Name	Color	Type	Dim	Tag
Stage 1 Face	default	PowerDensity	2	8,9
Stage 2 Face	cyan	PowerDensity	2	44,45
Stage 3 Face	red	PowerDensity	2	20,21
Stage 3 Wall	red	PowerDensity	2	23,24

FIGURE 199. Surface Selector dialog

7. Click the *Add Surface(s)* button beneath the table. This will bring up the graphics display window containing the geometry model.

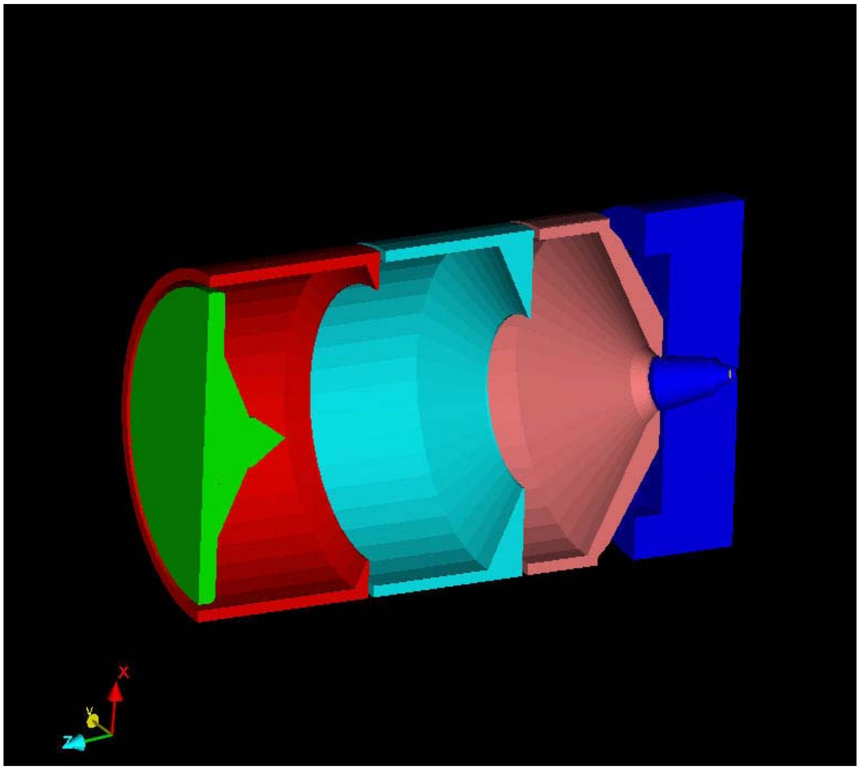


FIGURE 200. Geometry model oriented to show Stage 1, 2 and 3 Faces

8. Rotate the model so that you can see the face sections of Stages 1, 2 and 3, as shown in Figure 200.
9. Select the Stage 1 Face by pressing the Ctrl-Left mouse button. (The Stage 1 Face surface is a named surface in the CAD file consisting of tags 8 and 9.) The Stage 1 Face will appear selected (i.e., it will change from the default pink color to a dark green). Press Enter and the Stage 1 Face will appear in the *Selected Surfaces* table.
10. Next, add the Stage 2 Face surface. Again click the *Add Surface(s)* button beneath the table on the *Surface Selector* dialog. This will bring up the graphics display window containing the geometry model.
11. Select the Stage 2 Face by pressing the Ctrl-Left mouse button. (The Stage 2 Face surface is a named surface in the CAD file consisting of tags 44 and 45.) The Stage 2 Face will appear selected (it will change from cyan to a dark green). Press Enter and the Stage 2 Face will appear in the *Selected Surfaces* table.
12. Repeat the above procedure two more times to add the Stage 3 Face (a named CAD surface consisting of tags 20 & 21) and the Stage 3 Wall (a named CAD surface consisting of tags 23 & 24).
13. After you have added all four surfaces, the *Selected Surfaces* table should appear as shown in Figure 199. Click *OK* to close the *Surface Selector* dialog.

14. The power densities will appear on all four surfaces as shown in Figure 201.
15. Experiment with the settings in the *Power Densities (W/m^2)* section of the *Beam Optics Display Control Panel* dialog to see how the picture in the graphics display window changes. For example, increase the *No. of Contours* to get a more accurate look at power densities. Or highlight the regions of highest intensity by increasing the *Min* value for the range. Figure 202 shows the *No. of Contours* set to ‘400’ and the *Min* value set to ‘1000000’ W/m^2. Notice the region of highest intensity on the Stage 1 Face.

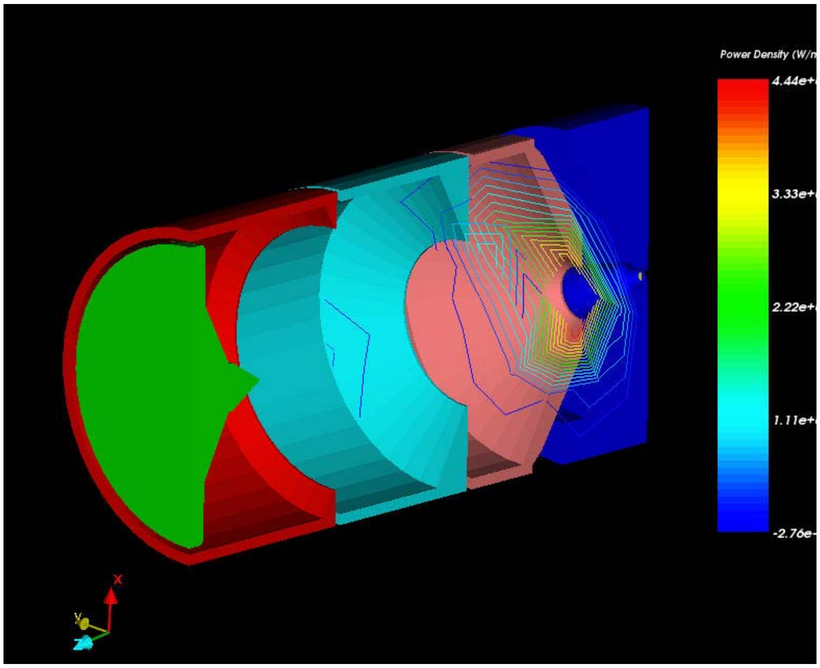


FIGURE 201. Power densities shown on four defined surfaces (20 contour

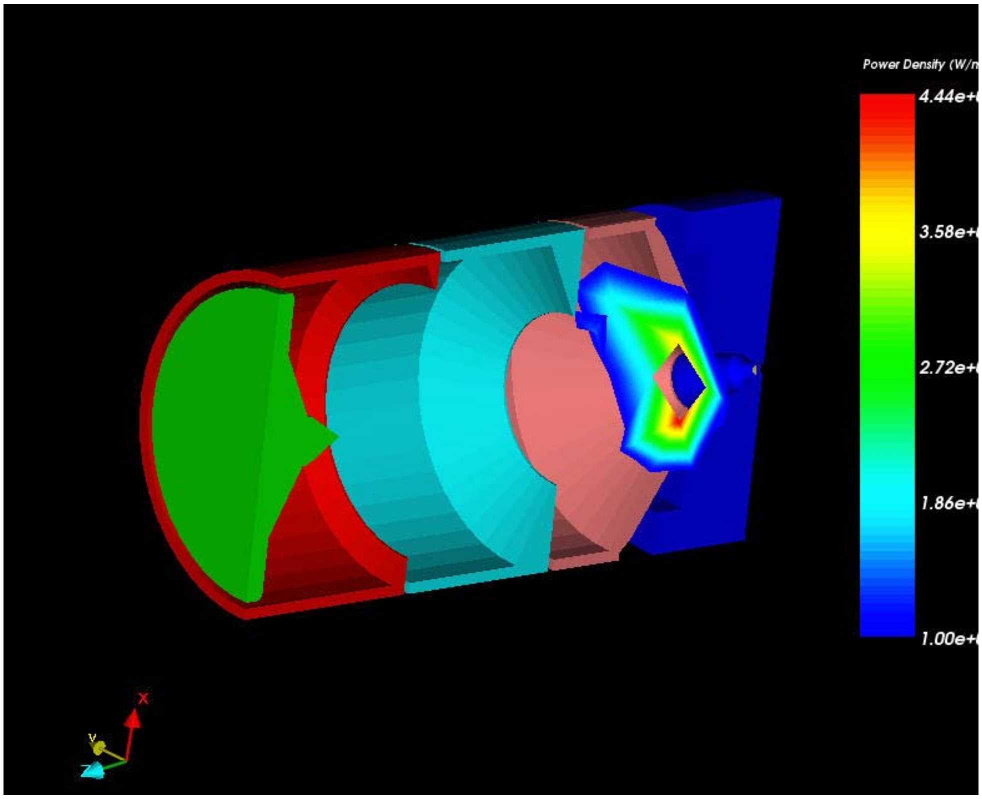


FIGURE 202. Power Densities shown on Stage 1, 2 and 3 Faces and Stage 3 Wall. (*No. of Contours* = 400 and *Min* value = 1000000)

4.2.1.12 Viewing Electrostatic Results

If desired, you can also view the Electrostatic results as you did in Examples 1A and 1B. Remember, the first time you click on *View Results->Electrostatics* you will be prompted to run the Electrostatic postprocessor.

4.2.1.13 Saving the Case

Before closing the case, make sure to save it. Right-click on the case name in the Project/Case Tree Control and select *Save Case*.

4.2.2 Example 2B - Electro/Magnetostatic Beam Analysis w/ Secondary Emissions

This example extends Example 2A by adding an external magnetic field and adding secondary emissions.

4.2.2.1 Creating a New Case

To begin, you will copy the previous Case, Example 2A, to a new Case, as follows.

1. Right click on the 'Case: Example 2A' case in the Project/Case Manager tree control. The submenu shown in Figure 203 will appear.
2. Select *Save Case As...* from the sub-menu. The *Copy Case Configurator* dialog will appear.
3. Change the name of the Case from 'Copy of Example 2A' to 'Example 2B' as shown in Figure 204.

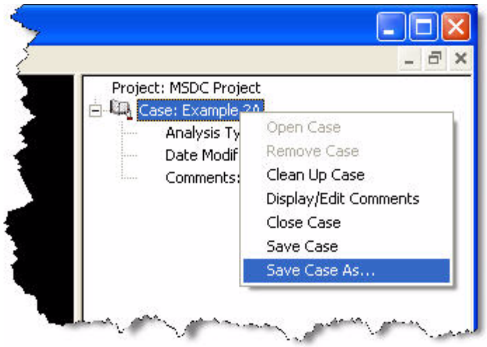
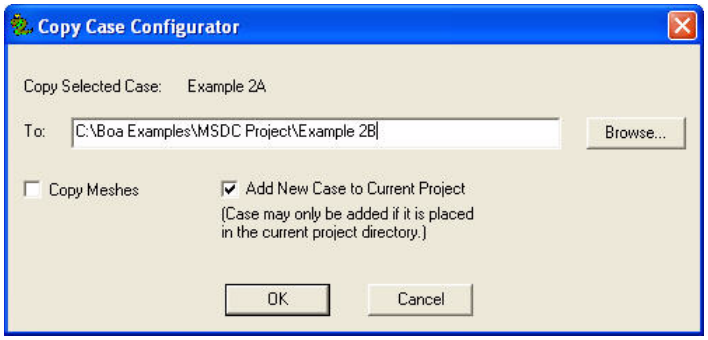

Project: MSDC Project
Case: Example 2A
Analysis Ty
Date Modif
Comments:
Open Case
Remove Case
Clean Up Case
Display/Edit Comments
Close Case
Save Case
Save Case As...

FIGURE 203. *Save Case As...* on the Case sub-menu


Copy Case Configurator
Browse...
OK
Cancel

Copy Selected Case: Example 2A

To: C:\Boa Examples\MSDC Project\Example 2B

☐ Copy Meshes

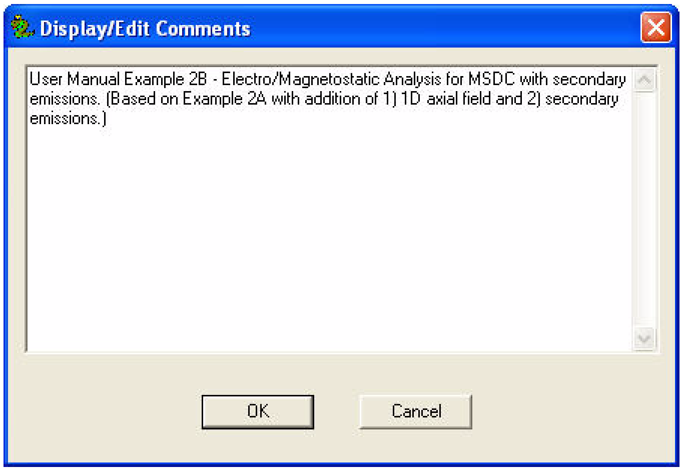
☑ Add New Case to Current Project

(Case may only be added if it is placed in the current project directory.)

FIGURE 204. Copy Case dialog

4. Uncheck the *Copy Meshes* checkbox. (You will generate a different initial mesh for this case.) Leave the other checkbox checked.
5. Click *OK*. The new case will appear in the Project/Case Manager tree control and the model will appear in the graphics display region.
6. To add comments for Example 2B, right click on the new case ('Case: Example 2B') in the Project/Case Manager tree control. Select *Display/Edit Comments* from the sub-menu. The *Display/Edit Comments* dialog will appear.
7. Modify the comments as shown in Figure 205.

8. Click *OK* to close the dialog.


Display/Edit Comments
OK
Cancel

User Manual Example 2B - Electro/Magnetostatic Analysis for MSDC with secondary emissions. (Based on Example 2A with addition of 1) 1D axial field and 2) secondary emissions.)

FIGURE 205. Display/Edit Comments dialog

4.2.2.2 Saving a Case

As you continue through this example, it is a good idea to save your work occasionally. To save a case, right click on the *Case* name in the Project/Case Manager tree control and select *Save Case*. (Alternatively, open the *File* pulldown menu and click *Save Case*.)

4.2.2.3 Defining Analysis Type

In Example 2B you will add an external magnetic field. So this is an *Electro/Magnetostatic Beam* analysis type.

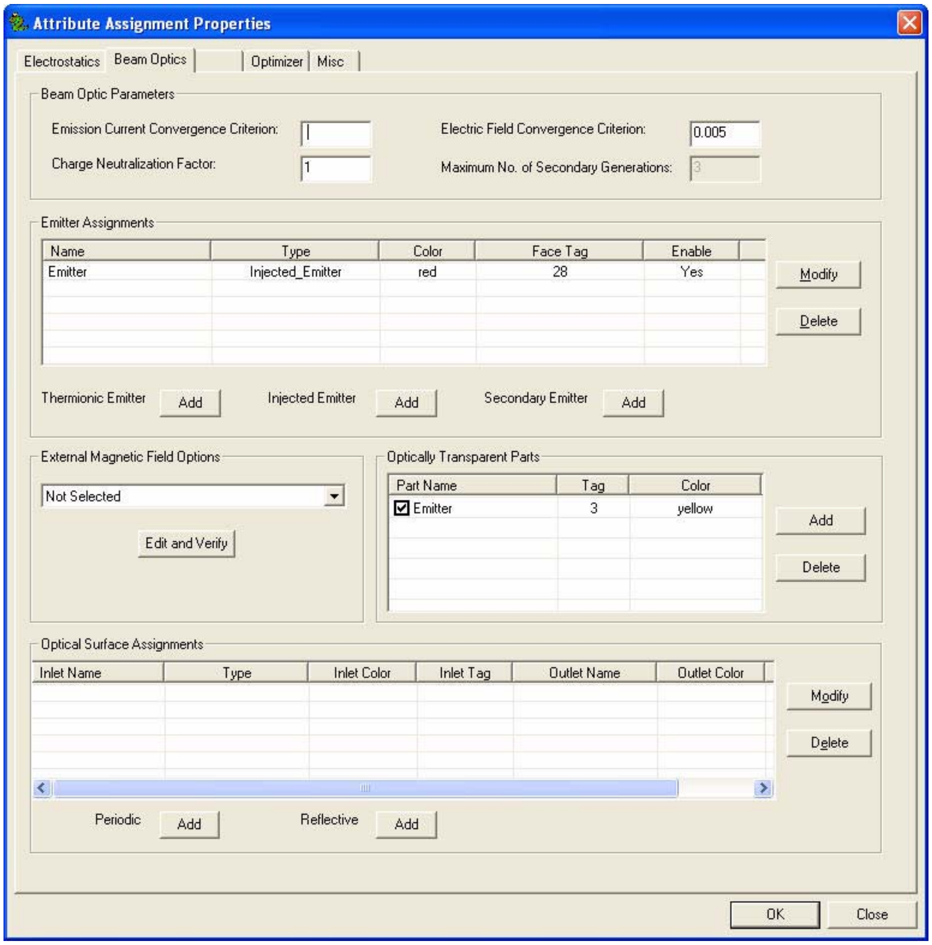
1. Open the *Analysis* pulldown menu and select *Electro/Magnetostatic Beam* analysis type.

4.2.2.4 Defining the External Magnetic Field

This example is based on Example 2A. Therefore the geometry file has been specified, the electrostatic attributes have been assigned and the injected emitter (a beam optics attribute) has been defined. Next you will add the external magnetic field.

1. Open the *Attribute Assignment Properties* dialog (shown in Figure 206) by clicking on the *Beam Optics* tab.

2. In the *External Magnetic Field Options* section, set the drop-down box to '1D Axial Imported Magnetic Flux Density'.


Attribute Assignment Properties
Electrostatics
Beam Optics
Optimizer
Misc
Beam Optic Parameters
Emitter Assignments
Modify
Delete
Thermionic Emitter
Add
Injected Emitter
Add
Secondary Emitter
Add
External Magnetic Field Options
Edit and Verify
Optically Transparent Parts
Add
Delete
Optical Surface Assignments
Modify
Delete
Periodic
Add
Reflective
Add
OK
Close

Emission Current Convergence Criterion:
Charge Neutralization Factor: 1
Electric Field Convergence Criterion: 0.005
Maximum No. of Secondary Generations: 3

Name	Type	Color	Face Tag	Enable
Emitter	Injected_Emitter	red	28	Yes

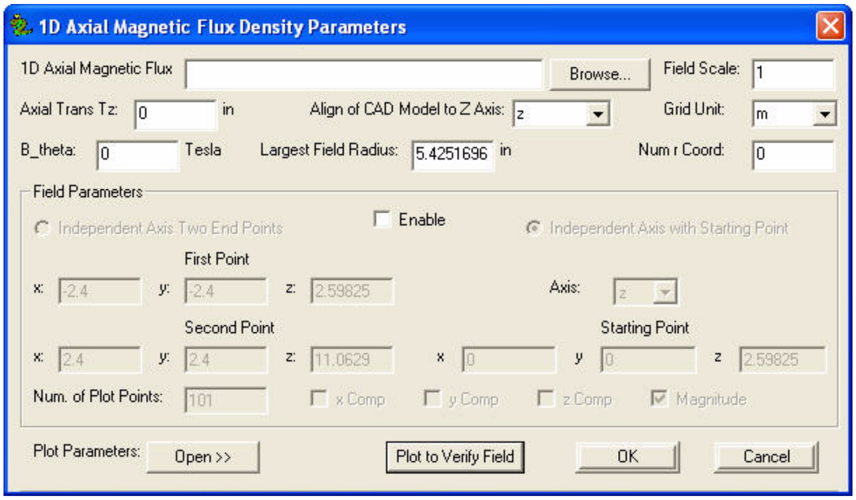
External Magnetic Field Options: Not Selected

Part Name	Tag	Color
☑ Emitter	3	yellow

Inlet Name	Type	Inlet Color	Inlet Tag	Outlet Name	Outlet Color

FIGURE 206. Attribute Assignment Properties dialog

3. The dialog in Figure 207 will pop up.



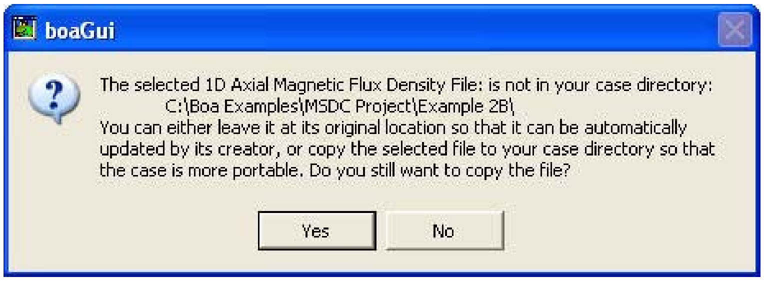
1D Axial Magnetic Flux Density Parameters

1D Axial Magnetic Flux:
Field Scale: 1
Axial Trans Tz: 0 in
Align of CAD Model to Z Axis: z
Grid Unit: m
B_theta: 0 Tesla
Largest Field Radius: 5.4251696 in
Num r Coord: 0

Field Parameters
○ Independent Axis Two End Points
☐ Enable
● Independent Axis with Starting Point
First Point x: -2.4 y: -2.4 z: 2.59825
Axis: z
Second Point x: 2.4 y: 2.4 z: 11.0629
Starting Point x: 0 y: 0 z: 2.59825
Num. of Plot Points: 101
☐ x Comp ☐ y Comp ☐ z Comp ☑ Magnitude

FIGURE 207. 1D Axial Magnetic Flux Density Parameters UPDATE (num r coord should not default to 0)

4. Use the *Browse* button next to the *1D Axial Magnetic Flux* entry box to locate the external magnetic field file named '_Bz.bf'. If you installed BOA using the default directory settings, this file will be located in C:/Program Files/CCR/Beam Optics Analyzer/Examples/Data Files. Select the file and click *Open* (or alternatively, double-click on the file).



boaGui

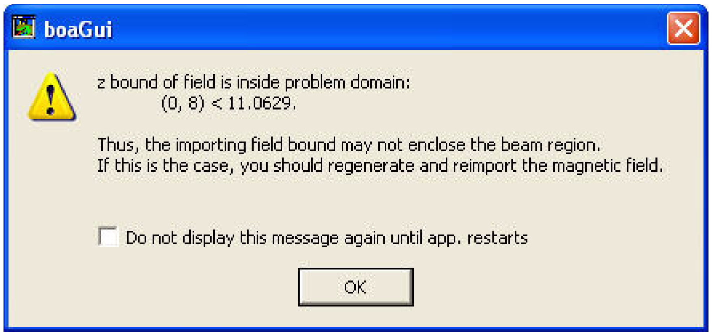
The selected 1D Axial Magnetic Flux Density File: is not in your case directory:
C:\Boa Examples\MSDC Project\Example 2B\
You can either leave it at its original location so that it can be automatically updated by its creator, or copy the selected file to your case directory so that the case is more portable. Do you still want to copy the file?

FIGURE 208. Copy File Message for the External Magnetic Field File

5. The message shown in Figure 208 will give you the option to create a copy of the external magnetic field file in the current Case directory. Click *Yes*.

6. Back on the *1D Axial Magnetic Flux Density Parameters* dialog, set the *Largest Field Radius* to '1.6' (half of the outer diameter of the collector), the *Grid Unit* to 'in' and the *Num r Coord* to '51'.

7. Before clicking *OK*, you can choose to visually inspect a plot of the magnetic field to confirm it is correct. Click the *Plot to Verify Field* button. The information message shown in Figure 209 will warn you that the z bound of the magnetic field is inside of the problem domain. In this case, this is expected because [Thuc please give me a short explanation of why this is expected behavior].



boaGui

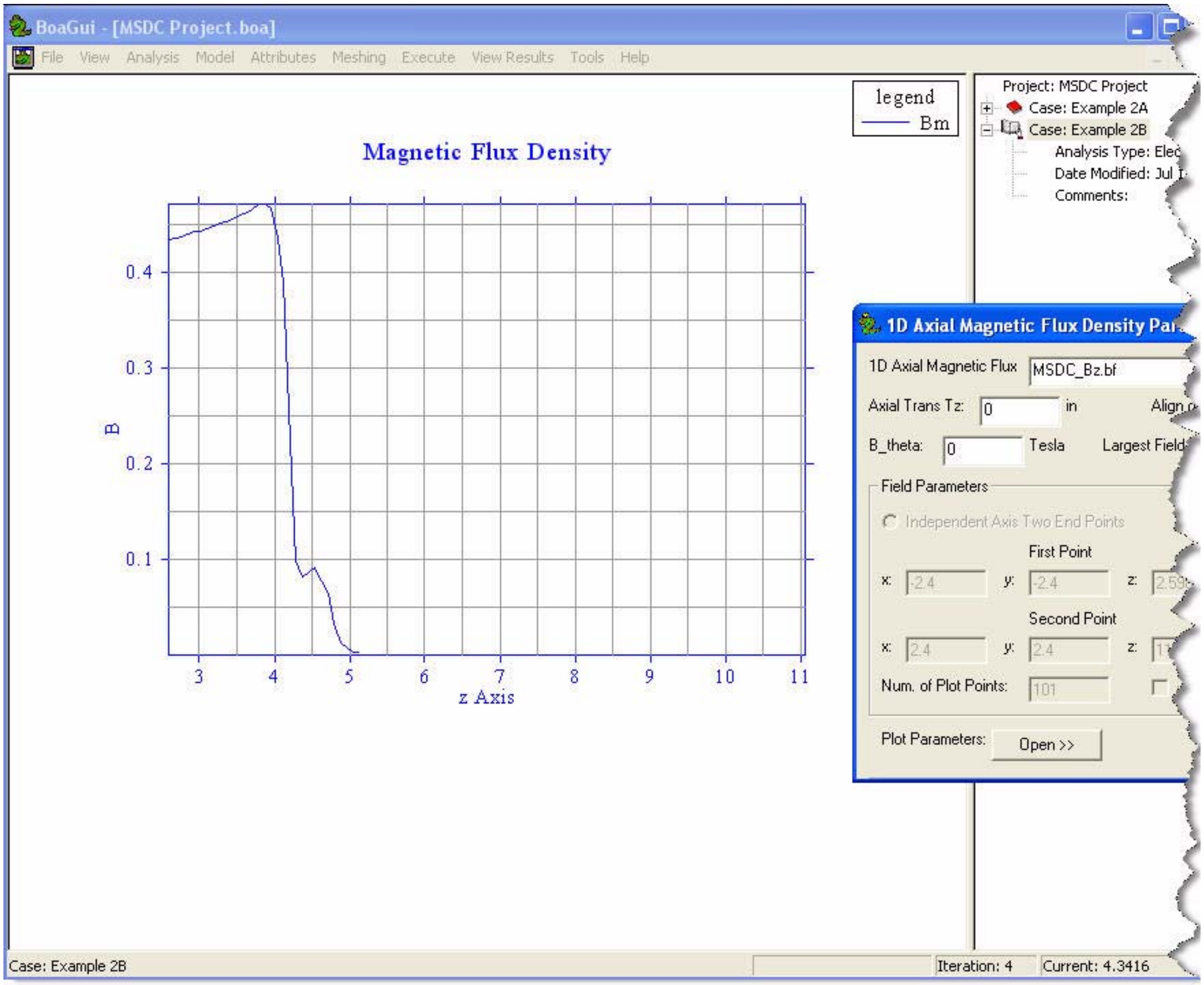
z bound of field is inside problem domain:
(0, 8) < 11.0629.

Thus, the importing field bound may not enclose the beam region.
If this is the case, you should regenerate and reimport the magnetic field.

☐ Do not display this message again until app. restarts

FIGURE 209. Information Message about the location of the magnetic field boundary within the problem domain UPDATE? Did this message change? Now includes x and y too

8. A plot of the external magnetic field will appear in the main window, as shown in Figure 210.


BoaGui - [MSDC Project.boa]
File
View
Analysis
Model
Attributes
Meshing
Execute
View Results
Tools
Help
legend
Bm
Magnetic Flux Density
0.4
0.3
0.2
0.1
B
3
4
5
6
7
8
9
10
11
z Axis
Project: MSDC Project
Case: Example 2A
Case: Example 2B
Analysis Type: Elec
Date Modified: Jul 1
Comments:
1D Axial Magnetic Flux Density Par
Align
Largest Field
Field Parameters
Independent Axis Two End Points
First Point
Second Point
Plot Parameters:
Open >>

1D Axial Magnetic Flux: MSDC_Bz.bf

Axial Trans Tz: 0 in

B_theta: 0 Tesla

First Point x: -2.4 y: -2.4 z: 2.59

Second Point x: 2.4 y: 2.4 z: 11

Num. of Plot Points: 101

Case: Example 2B

Iteration: 4

Current: 4.3416

FIGURE 210. Plot of the 1D Axial Magnetic Flux Density

9. When you are satisfied that the external magnetic field is correct, click *OK* on the *1D Axial Magnetic Flux Density Parameters* dialog to close the plot. The main window will return to the geometry model and the*1D Axial Magnetic Flux Density Parameters* dialog will close.

4.2.2.5 Defining the Secondary Emitters

Next you will define the secondary emitters

1. The *Attribute Assignment Parameters* dialog should still be open, with the *Beam Optics* tab active.
2. In the *Emitter Assignments* section, click the *Secondary Emitter Add* button beneath the table. This will bring up the graphics display window containing the geometry model.
3. In order to specify the secondary emitter surfaces, you will first need to slice the model to expose the interior of the collector. Open the *View* menu and select *Slice Model*. On the resulting *Slice Electrostatic Model* dialog, select the 'zx plane' in the *Choose Plane* section. Click *Close* to close the *Slice Electrostatic Model* dialog.

4. Rotate the model so that you can see the face sections of Stages 1, 2 and 3, as shown in Figure 211.
5. Select the Stage 1 Face by pressing the Ctrl-Left mouse button. (The Stage 1 Face surface is a named surface in the CAD file consisting of tags 8 and 9.) The Stage 1 Face will appear selected (i.e., it will change from the default pink color to a dark green). Press Enter and the *Assign Secondary Electron Emission Parameters* dialog will appear (shown in Figure 212).
6. On the *Assign Secondary Electron Emission Parameters* dialog, change the *Color* to ‘Yellow’ and leave all other parameters at their default values (*Number of Secondaries* = ‘3’, *Surface Material* = ‘Copper’ and the secondary model set to ‘Atomic Specification’).
7. Click *OK* to close the dialog.
8. The new secondary emitter will appear as the second entry in the *Emitter Assignments* table on the *Assign Attributes Properties* dialog, as shown in Figure 213. (Figure 213 shows the *Emitter Assignments* table with four second-

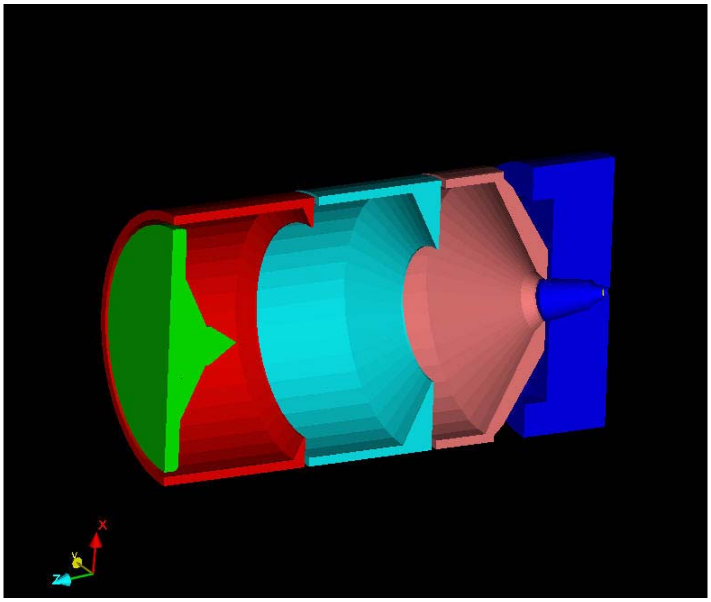
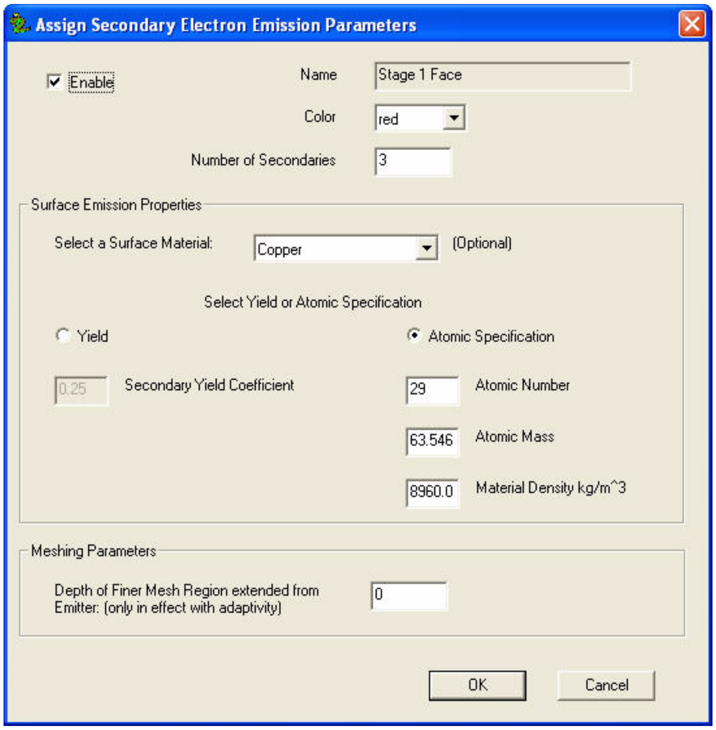


FIGURE 211. Geometry model oriented to show Stage 1, 2 and 3 Faces


Assign Secondary Electron Emission Parameters

☑ Enable

Name: Stage 1 Face

Color: red

Number of Secondaries: 3

Surface Emission Properties

Select a Surface Material: Copper (Optional)

Select Yield or Atomic Specification

○ Yield ● Atomic Specification

0.25 Secondary Yield Coefficient

29 Atomic Number

63.546 Atomic Mass

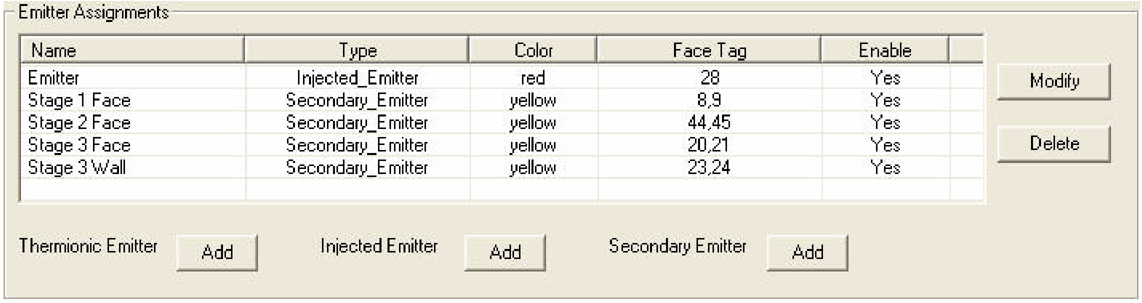
8960.0 Material Density kg/m^3

Meshing Parameters

Depth of Finer Mesh Region extended from Emitter: (only in effect with adaptivity): 0

FIGURE 212. Assign Secondary Electron Emission Parameters dialog

ary emitters. This is how the table will appear after you have completed all secondary emitter definitions, as described in the next few steps.)



Name	Type	Color	Face Tag	Enable
Emitter	Injected_Emitter	red	28	Yes
Stage 1 Face	Secondary_Emitter	yellow	8,9	Yes
Stage 2 Face	Secondary_Emitter	yellow	44,45	Yes
Stage 3 Face	Secondary_Emitter	yellow	20,21	Yes
Stage 3 Wall	Secondary_Emitter	yellow	23,24	Yes

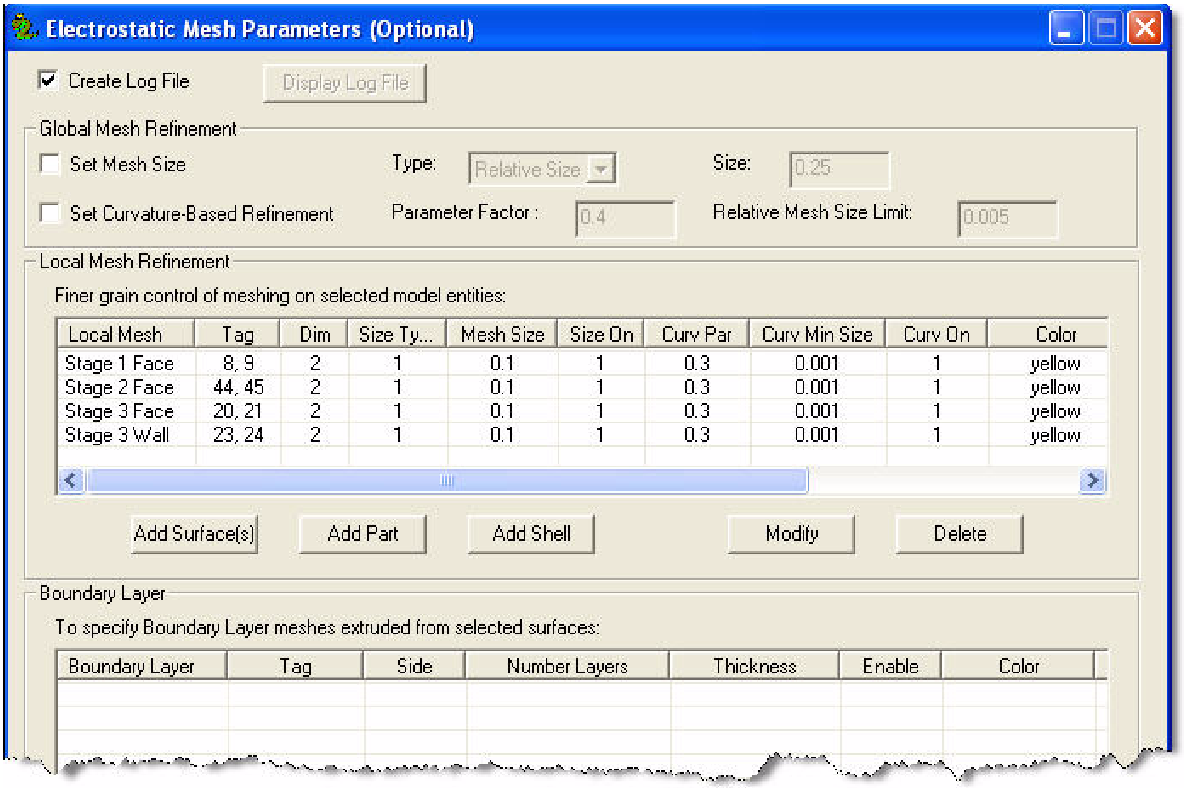
FIGURE 213. Emitter Assignments table shown with four Secondary Emitters defined

9. Next, define the Stage 2 Face as a secondary emitter. Again, click the *Secondary Emitter Add* button beneath the *Emitter Assignments* table. This will bring up the graphics display window containing the geometry model.
10. Select the Stage 2 Face by pressing the Ctrl-Left mouse button. (The Stage 2 Face surface is a named surface in the CAD file consisting of tags 44 and 45.) The Stage 2 Face will appear selected (it will change from cyan to a dark green). Press Enter and the *Assign Secondary Electron Emission Parameters* dialog will reappear.
11. Again change the *Color* to ‘Yellow’ and leave all other parameters at their default values.
12. Click *OK* to close the dialog. The new secondary emitter will appear as the third entry in the *Emitter Assignments* table (Figure 213) on the *Assign Attributes Properties* dialog.
13. Repeat the above procedure two more times to define Stage 3 Face (a named CAD surface consisting of tags 20 & 21) and the Stage 3 Wall (a named CAD surface consisting of tags 23 & 24) as secondary emitters.
14. After you have added all four secondary emitters, the *Emitter Assignments* table should appear as shown in Figure 213. (Note: As discussed in the next section, BoaGui will automatically generate *Local Mesh Refinements* for each of these surfaces. (It is important to model these surfaces with a higher degree of accuracy since they are now defined as secondary emitters.)
15. Click *OK* to close the *Attribute Assignment Properties* dialog.
16. {Thuc, sometimes when I close the *Attribute Assignment Properties* dialog, the geometry model appears a little odd. See separate email message. Generating the Initial Electrostatic Mesh} check if still happening

Once you complete the attribute assignments, you are ready to generate the initial electrostatic mesh.

1. Open the *Meshing* pulldown menu and select *Parameters.*

2. The *Electrostatic Mesh Parameters (Optional)* dialog will appear, as shown in Figure 214. Notice that *Local Mesh Refinements* table contains four entries that you did not explicitly create. BoaGui automatically generated these entries -- one local mesh refinement for each of the secondary emitter surfaces that you defined in the previous section. (BOAGui auto-generates these entries because is important to model secondary emitter surfaces more accurately.)



☑ Create Log File

Global Mesh Refinement

☐ Set Mesh Size   Type: Relative Size   Size: 0.25

☐ Set Curvature-Based Refinement   Parameter Factor : 0.4   Relative Mesh Size Limit: 0.005

Local Mesh Refinement

Finer grain control of meshing on selected model entities:

Local Mesh	Tag	Dim	Size Ty...	Mesh Size	Size On	Curv Par	Curv Min Size	Curv On	Color
Stage 1 Face	8, 9	2	1	0.1	1	0.3	0.001	1	yellow
Stage 2 Face	44, 45	2	1	0.1	1	0.3	0.001	1	yellow
Stage 3 Face	20, 21	2	1	0.1	1	0.3	0.001	1	yellow
Stage 3 Wall	23, 24	2	1	0.1	1	0.3	0.001	1	yellow

Boundary Layer

To specify Boundary Layer meshes extruded from selected surfaces:

Boundary Layer	Tag	Side	Number Layers	Thickness	Enable	Color

FIGURE 214. Local Mesh Refinements table shown with four automatically generated entries (corresponding to the four secondary emitters)

3. No changes are needed to the meshing parameters, so click *OK* to close the dialog. Click
4. Reopen the *Meshing* menu and select *Generate Initial Mesh*.
5. A DOS window will appear indicating progress during the mesh generation process. When the process completes, the following text will appear at the bottom of the window: “Mesh Generation Complete, mesh file created: _E.sms !Press any key to continue/exit...”. Hit any key to close the DOS window and return control back to the user interface.

(If desired, you can view the mesh by opening the *Meshing* pulldown menu and selecting *Display Initial Mesh*. The mesh will appear in the graphics display region. This step was illustrated in the previous JLAB example 1A. See Example 1A for details.)

4.2.2.6 Executing BOA

Once the initial mesh is generated, you can execute the BOA engine.

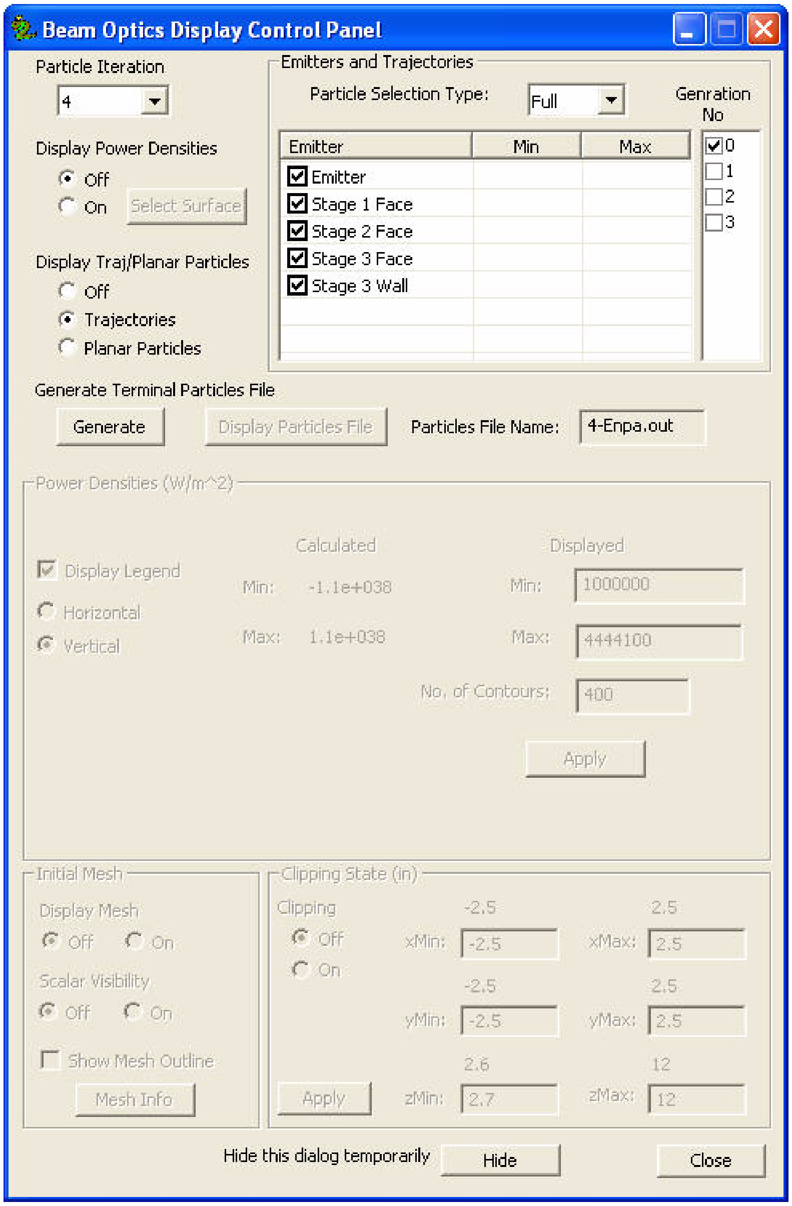
1. Open the *Execute* pulldown menu and select *Single Case->Execute BOA*.

2. Leave all parameters set to the default values. (Note that *Max Number of Adaptivity Passes* is set to '0'which means the mesh will not be adapted for this run.) Click *Execute*.
3. A DOS window will appear indicating progress during BOA execution. When BOA completes, the following text will appear at the bottom of the window "Proceed to View Results. Press any key to continue/exit". Hit any key to close the DOS window and return control back to the user interface. (Expect this run to take longer than the previous run.)

4.2.2.7 Viewing Beam Analysis Results

For this example, you will proceed directly to the Beam Analysis results to view the trajectories for the three different generations of secondaries.

1. Open the *View Results* pulldown menu and select *Beam Analysis*. The *Beam Optics Display Control Panel* dialog shown in Figure 215 will appear.
2. By default, the primary trajectories are shown. (The *Display Traj/Planar Particles* radio button is set to display 'Trajectories' and the *Generation No* checkboxes in the *Emitters and Trajectories* section show only the '0' generation checked.)



Beam Optics Display Control Panel

Particle Iteration: 4

Display Power Densities: ● Off ○ On

Display Traj/Planar Particles: ○ Off ● Trajectories ○ Planar Particles

Emitters and Trajectories

Particle Selection Type: Full

Emitter	Min	Max
☑ Emitter		
☑ Stage 1 Face		
☑ Stage 2 Face		
☑ Stage 3 Face		
☑ Stage 3 Wall		

Genration No: ☑ 0 ☐ 1 ☐ 2 ☐ 3

Generate Terminal Particles File

Particles File Name: 4-Enpa.out

Power Densities (W/m^2)

☑ Display Legend ○ Horizontal ● Vertical

	Calculated	Displayed
Min:	-1.1e+038	1000000
Max:	1.1e+038	4444100

No. of Contours: 400

Initial Mesh

Display Mesh: ● Off ○ On

Scalar Visibility: ● Off ○ On

☐ Show Mesh Outline

Clipping State (in)

Clipping: ● Off ○ On

xMin: -2.5 xMax: 2.5

yMin: -2.5 yMax: 2.5

zMin: 2.7 zMax: 12

FIGURE 215. Beam Optics Display Control Panel dialog

3. To show the trajectories for different generations of secondaries, check the desired *Generation No* in the *Emitters and Trajectories* section. For example, Figure 216 shows the primaries and the first generation of secondaries. (The *Generation No* checkboxes are checked for ‘0’ and ‘1’.)
4. Figure 218 shows the primaries and the first and second generation of secondaries. (The *Generation No* checkboxes are checked to include ‘0’, ‘1’ and ‘2’.)

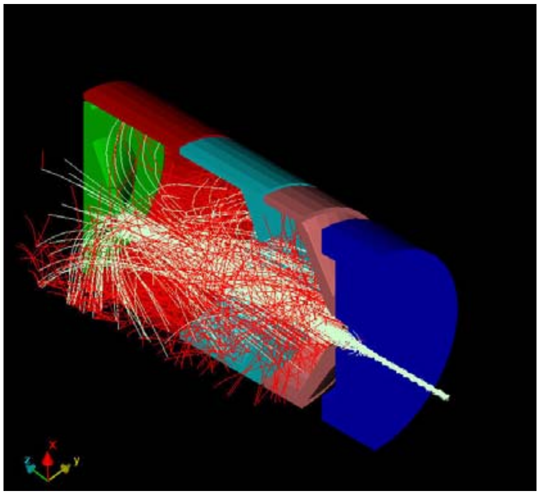


FIGURE 216. Primaries and First Generation of Secondaries

5. Figure 218 shows the primaries and the first, second and third generation of secondaries. (The *Generation No* checkboxes are checked to include ‘0’, ‘1’, ‘2’ and ‘3’.)
6. You can also turn different emitter surfaces on and off using the checkboxes in front of the *Emitter* names in the Emitter table. Play around with these controls to see how the display changes. Zoom in and out to get a closer look at the trajectories.

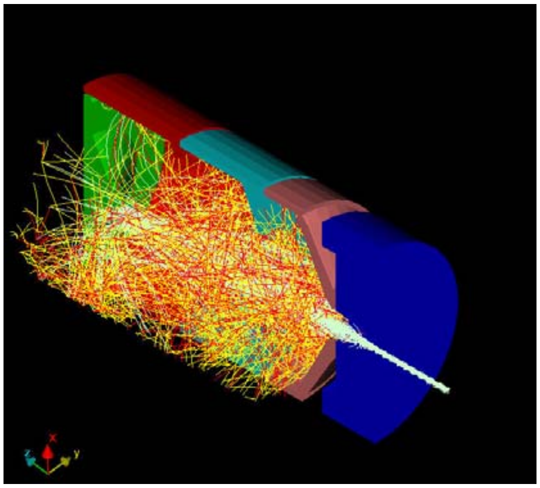


FIGURE 217. Primaries and First & Second Generation of Secondaries

7. If desired, you can also change the color of the trajectories. Open the *View* menu and select *Color->Trajectories*. On the resulting *Color Trajectories* dialog, select the desired generation and click *Change Color.* The *Color* palette dialog will pop up. Select a color and click *OK* to change the color and close the *Color* palette. To close the *Color Trajectories* dialog, click the *Close* button.

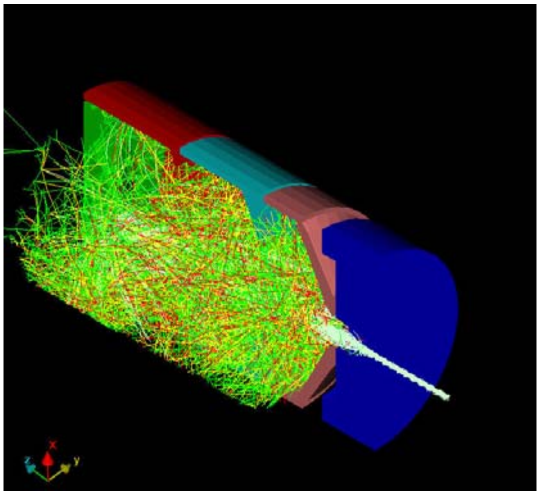


FIGURE 218. Primaries and First, Second & Third Generation of Secondaries

4.2.2.8 Viewing Electrostatic Results

You can also view the Electrostatic results as you did in Examples 1A and

1B. Remember, when you click on *View Results->Electrostatics* the first time you will be prompted to run the Electrostatic postprocessor.

4.2.2.9 Saving the Case

Before closing the case, make sure to save it. Right-click on the case name in the Project/Case Tree Control and select *Save Case*.

4.3 Example 3 - Solenoid for a Multibeam Klystron (MBK)

4.3.1 Example 3A - Magnetostatic Analysis

[revise to match format of previous examples]

The geometry is a solenoid for a multibeam klystron (MBK). This klystron requires a uniform field from the gun through the RF circuit, but a diverging field after the circuit to spread the beam into the collector. The return has eight holes for the beam to pass through in the collector polepiece, shown at the right. There is one hole on center at the gun (left) end. This is for access to the cathode. The model is shown in Figure 219.

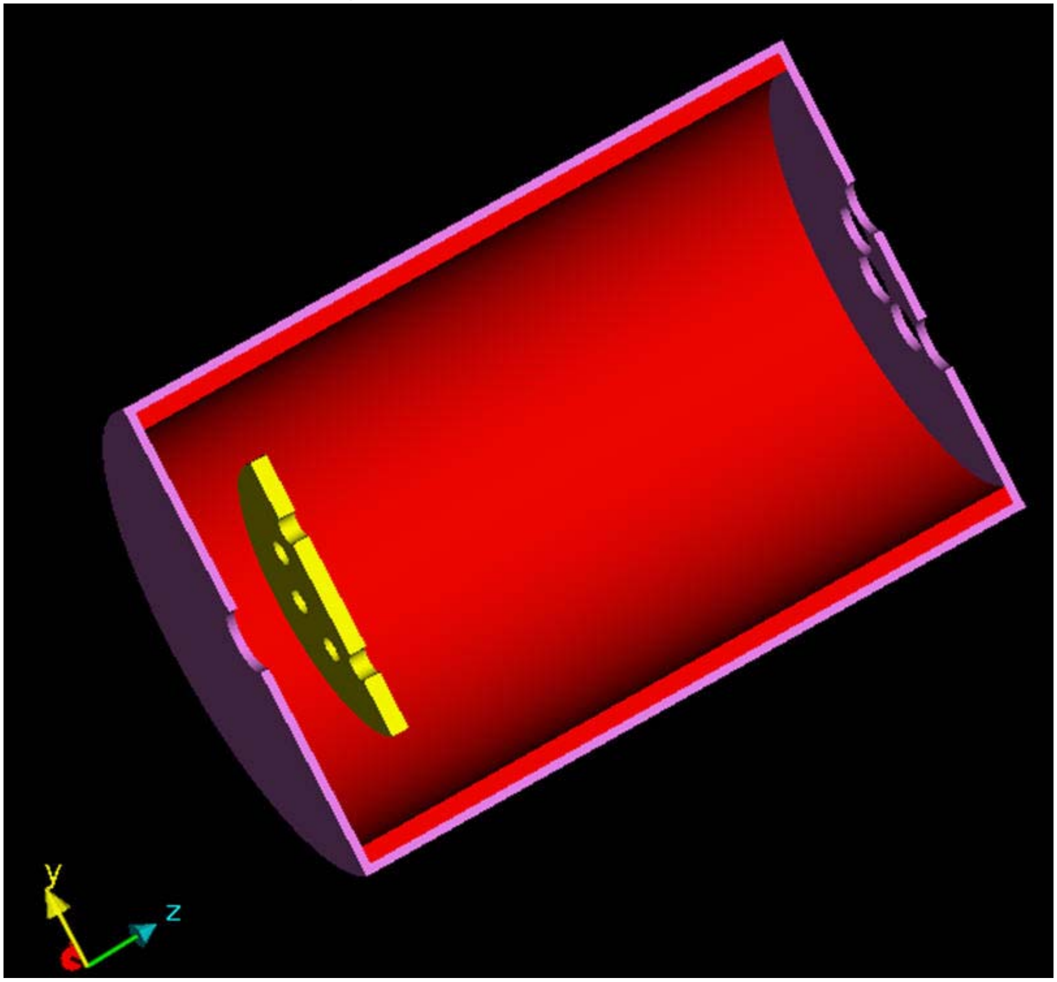
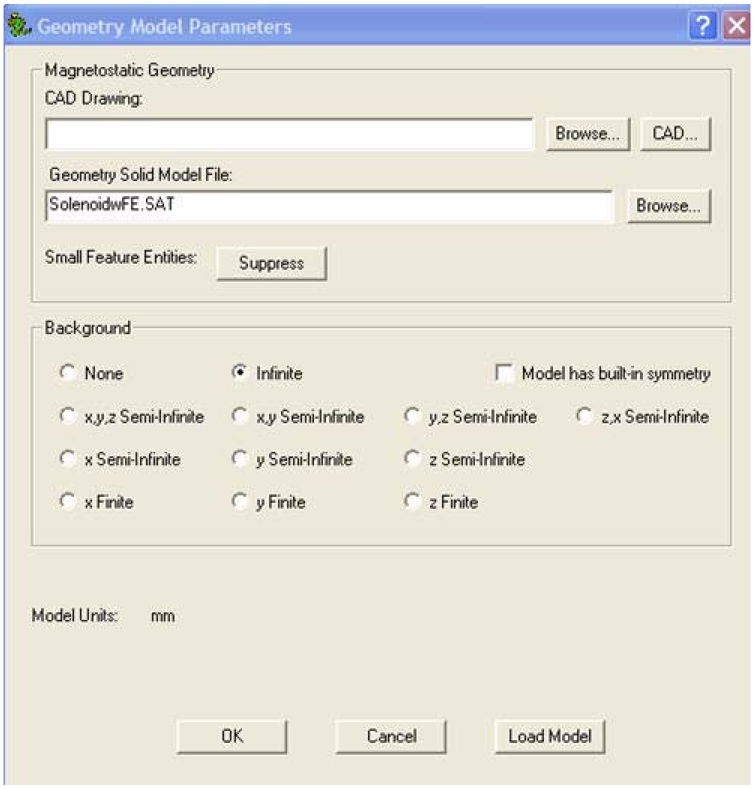


FIGURE 219. Model for the MBK solenoid. The coil is red and the iron return is violet. The yellow object is the focus electrode for the electron gun. This is not part of the magnetics problem, and will be ignored by BOA. It is included to show how parts from other relevant structures can be included for reference.

The problem was started by selecting *Magnetostatic* from the drop down menu under *Analysis*. Next, an ACIS (.sat) file from a solid modeling program was imported using the dialog activated by selecting *Model* in menu, and shown in Figure 220. In this case, the program was Solid Works. For this simple model, there were no small features that were likely to cause problems with meshing, so the *Small Feature Entities* value was left at the default of 0. While the problem has symmetry around the y-z plane, it is simple enough to allow the whole geometry to be modeled within a reasonable time. Thus, the *Infinite* radio button on the *Background* selection portion of the dialog has been selected. The model units were identified by BOA as 'mm'. Hitting *OK* or *Load Model* loads the model.



Geometry Model Parameters

Magnetostatic Geometry

CAD Drawing:

Geometry Solid Model File: SolenoidwFE.SAT

Small Feature Entities:

Background

○ None ● Infinite ☐ Model has built-in symmetry

○ x,y,z Semi-Infinite ○ x,y Semi-Infinite ○ y,z Semi-Infinite ○ z,x Semi-Infinite

○ x Semi-Infinite ○ y Semi-Infinite ○ z Semi-Infinite

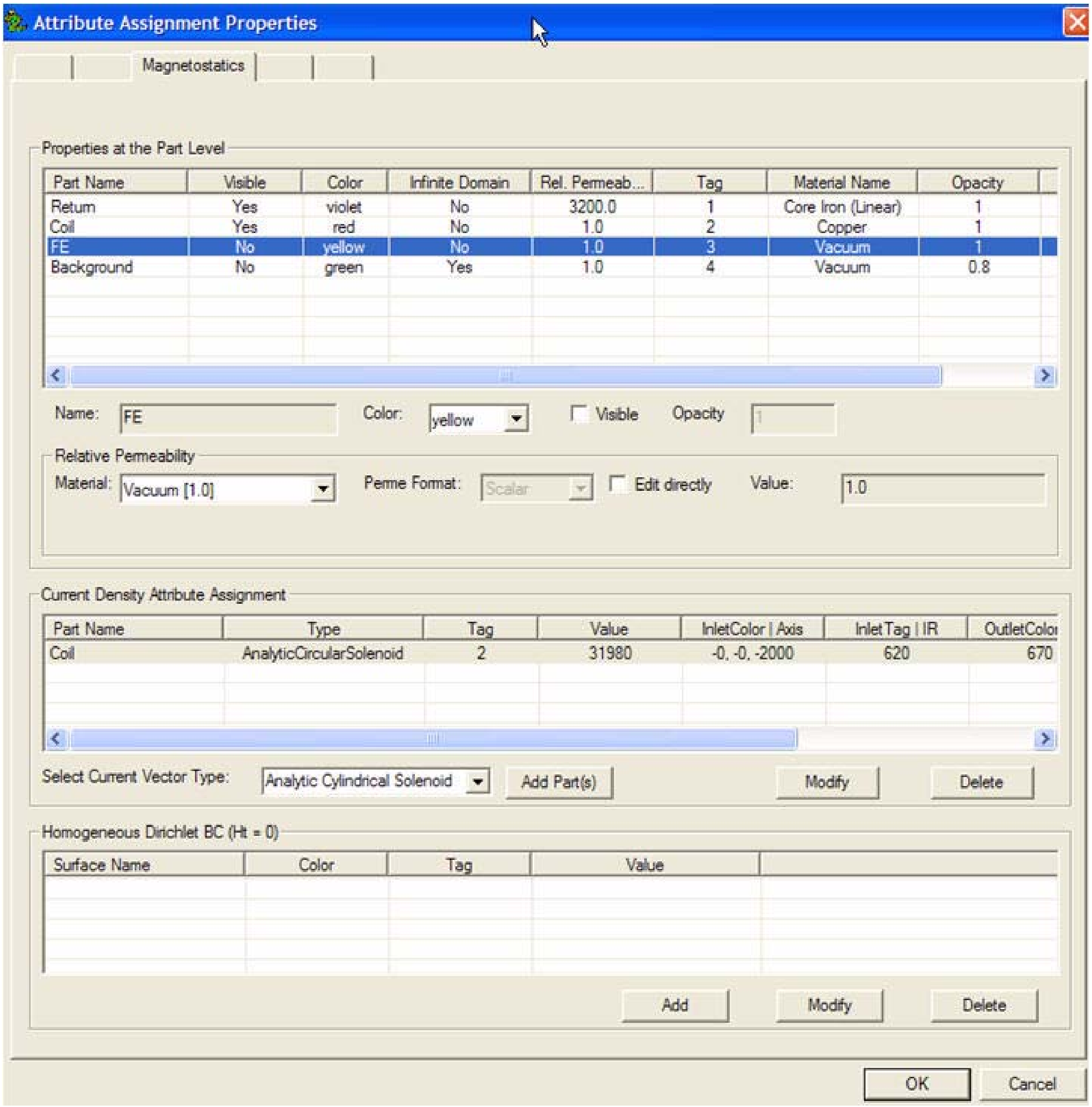
○ x Finite ○ y Finite ○ z Finite

Model Units: mm

FIGURE 220. Model GUI used to import the model.

The resulting graphic, after slicing (*View/slice model*) the model on the yz plane, is shown in Figure 219.

The next step is to assign attributes by opening the *Attributes Assignment Properties* dialog (see Figure 221). The return was initially assigned as 'iron' with a fixed permeability of 3200. We shall check the validity of the this assignment later. The coil was assigned as 'copper', but it could also be left as 'vacuum' for this problem where only the permeability is of importance. The 'FE' is the gun part that is included for reference. Its material is set to 'vacuum'.



Properties at the Part Level

Part Name	Visible	Color	Infinite Domain	Rel. Permeab...	Tag	Material Name	Opacity
Return	Yes	violet	No	3200.0	1	Core Iron (Linear)	1
Coil	Yes	red	No	1.0	2	Copper	1
FE	No	yellow	No	1.0	3	Vacuum	1
Background	No	green	Yes	1.0	4	Vacuum	0.8

Name: FE
Color: yellow
☐ Visible
Opacity: 1

Relative Permeability
Material: Vacuum [1.0]
Perme Format: Scalar
☐ Edit directly
Value: 1.0

Current Density Attribute Assignment

Part Name	Type	Tag	Value	InletColor | Axis	InletTag | IR	OutletColor
Coil	AnalyticCircularSolenoid	2	31980	-0, -0, -2000	620	670

Select Current Vector Type: Analytic Cylindrical Solenoid

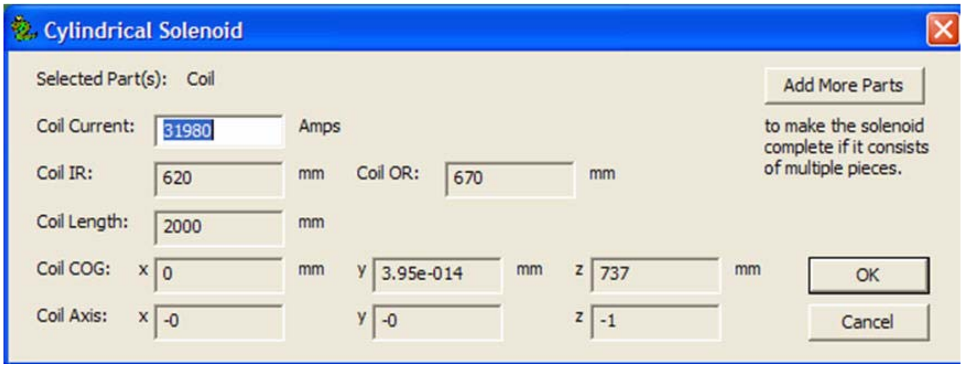
Homogeneous Dirichlet BC (Ht = 0)

Surface Name	Color	Tag	Value	

FIGURE 221. Attribute Assignment Properties dialog

The middle part of the dialog contains the means for assigning the coil currents. In this case, we have a simple solenoidal coil, and 'Analytic Cylindrical Solenoid' was selected as the *Current Vector Type*. The coil was selected by pushing *Add Parts*, the selecting the appropriate part on the model graphic by, as usual, holding the control key down while clicking the left mouse button. Hit return to exit the model graphics screen and bring up the *Cylindrical Solenoid* dialog shown in Figure 222. All of the entries except the coil current (highlighted) are filled in by BOA. The current must be specified. Our solenoid consists of a single part, and thus the dialog should be exited at this point by selecting *OK*.

The final section of the dialog is used to select surfaces to serve as Dirichlet boundaries. However, when importing the model, we selected quasi-infinite boundary conditions, and thus no entries are required in this part of the dialog.

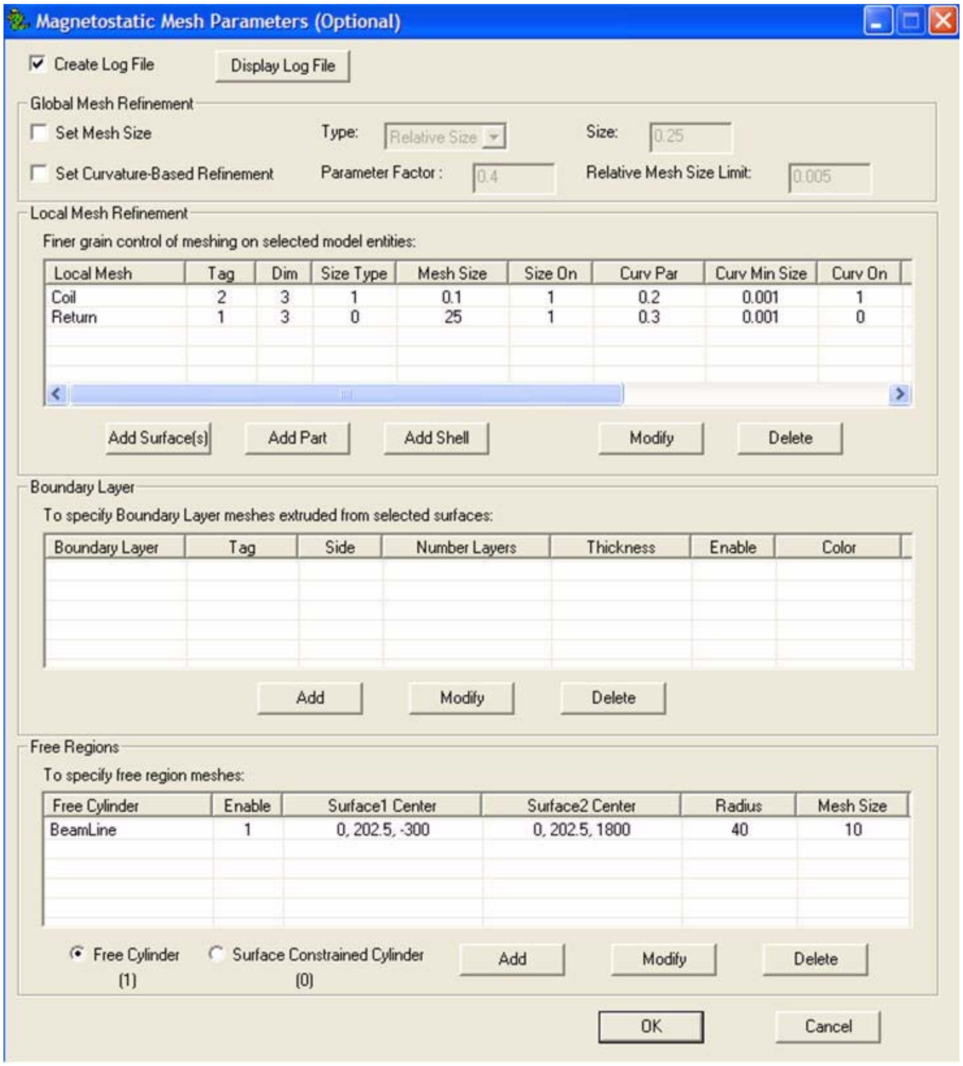

Cylindrical Solenoid
Add More Parts
to make the solenoid complete if it consists of multiple pieces.
OK
Cancel

Selected Part(s): Coil
Coil Current: 31980 Amps
Coil IR: 620 mm Coil OR: 670 mm
Coil Length: 2000 mm
Coil COG: x 0 mm y 3.95e-014 mm z 737 mm
Coil Axis: x -0 y -0 z -1

FIGURE 222. GUI for specifying the solenoid parameters.

Hit *OK* exit the dialog.

The next step is to mesh the model. Select *Meshing/ Parameters* from the menu, and the dialog shown in Figure 223 will appear. For this problem, only the *Local Mesh Refinement* and the *Free Regions* portions of the dialog will be used.



☑ Create Log File

Global Mesh Refinement

☐ Set Mesh Size Type: Relative Size Size: 0.25

☐ Set Curvature-Based Refinement Parameter Factor : 0.4 Relative Mesh Size Limit: 0.005

Local Mesh Refinement

Finer grain control of meshing on selected model entities:

Local Mesh	Tag	Dim	Size Type	Mesh Size	Size On	Curv Par	Curv Min Size	Curv On
Coil	2	3	1	0.1	1	0.2	0.001	1
Return	1	3	0	25	1	0.3	0.001	0

Boundary Layer

To specify Boundary Layer meshes extruded from selected surfaces:

Boundary Layer	Tag	Side	Number Layers	Thickness	Enable	Color

Free Regions

To specify free region meshes:

Free Cylinder	Enable	Surface1 Center	Surface2 Center	Radius	Mesh Size
BeamLine	1	0, 202.5, -300	0, 202.5, 1800	40	10

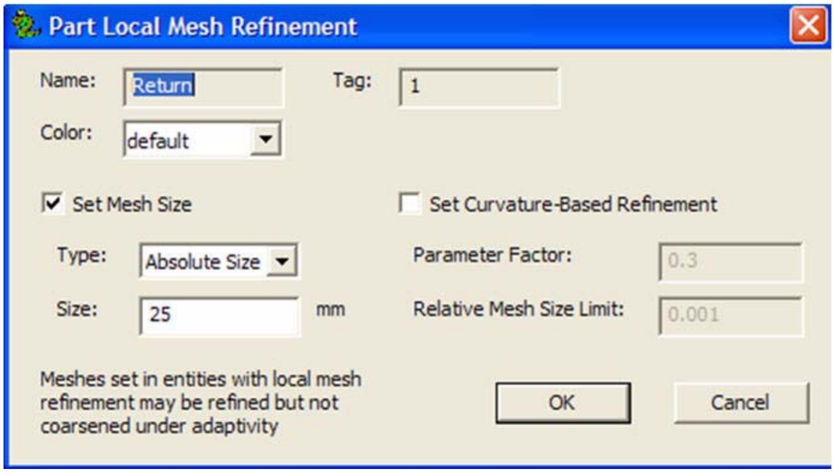
● Free Cylinder (1) ○ Surface Constrained Cylinder (0)

FIGURE 223. Mesh Parameters dialog.

In *Local Mesh Refinement*, BOA will have supplied the first entry ‘Coil’. This provides enhanced meshing of the coil. The return, since it is relatively thin, should also be meshed in more detail than the default. To do this, push *Add Part*, then select the return on the graphics screen with the usual cntrl/ left mouse button. Hit return to exit. The dialog shown in Figure 224 will appear. Make sure the *Set Mesh Size* box is checked. Choose ‘Absolute Size’ in the drop down menu for *Type* and enter 25 mm, the thickness of the return, for the value. Uncheck the *Set Curvature-Based Refinement* box, and hit *OK*.

Changes to this or the coil's meshing can be made by selecting the appropriate part in the table, and clicking on *Modify.*

For this problem, the only other entry in this dialog is a *Free Region*. With this feature, the user can specify a cylinder with a specified mesh density. This is often required to get an adequately smooth solution in a region which will have an electron beam. In this case, the cylinder is specified by selecting *Add* a free cylinder, and filling in the dialog shown in Figure 225. The parameters for the cylinder were chosen so that it encompassed the beam for the MBK.



Name: Return
Tag: 1
Color: default

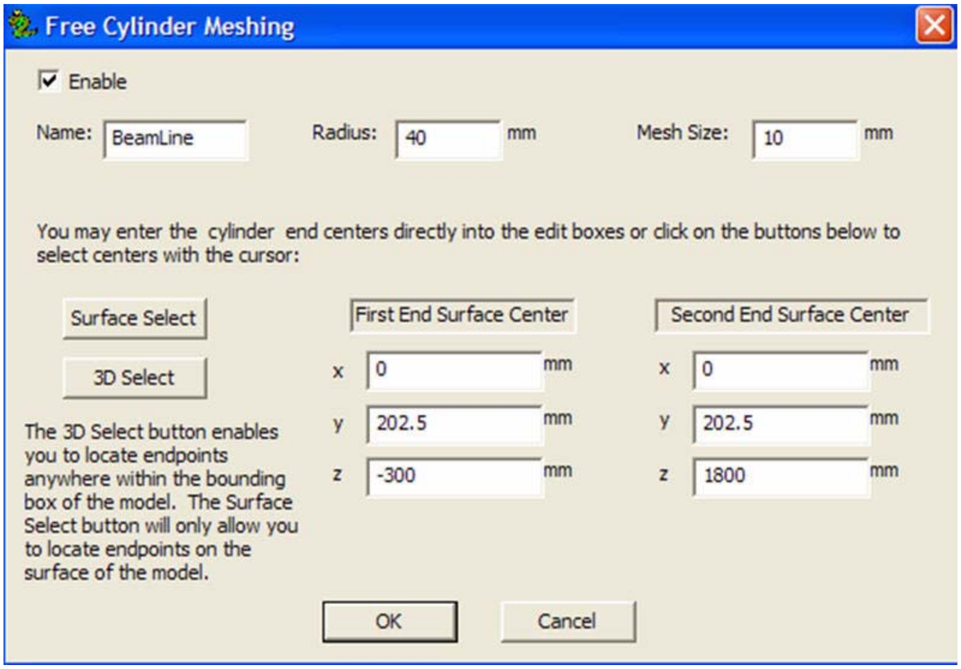
☑ Set Mesh Size
☐ Set Curvature-Based Refinement

Type: Absolute Size
Size: 25 mm
Parameter Factor: 0.3
Relative Mesh Size Limit: 0.001

Meshes set in entities with local mesh refinement may be refined but not coarsened under adaptivity

FIGURE 224. Part Local Mesh Refinement dialog.

The mesh can now be generated (*Mesh/ Generate Initial Mesh*) and viewed in the usual manner.



☑ Enable

Name: BeamLine
Radius: 40 mm
Mesh Size: 10 mm

You may enter the cylinder end centers directly into the edit boxes or click on the buttons below to select centers with the cursor:

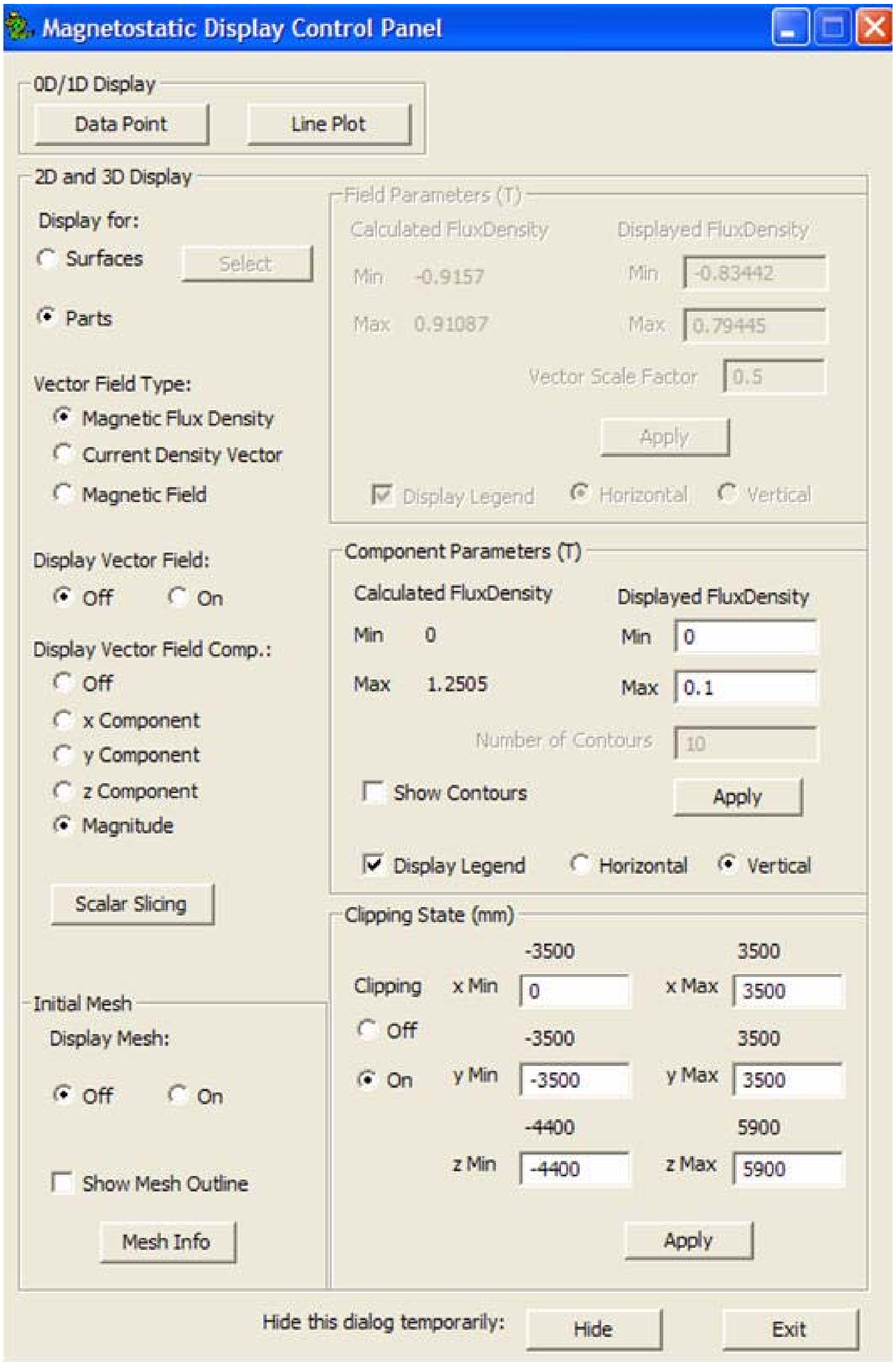
	First End Surface Center	Second End Surface Center
x	0 mm	0 mm
y	202.5 mm	202.5 mm
z	-300 mm	1800 mm

The 3D Select button enables you to locate endpoints anywhere within the bounding box of the model. The Surface Select button will only allow you to locate endpoints on the surface of the model.

FIGURE 225. Free cylinder definition GUI.

Next, the problem should run using *Execute/Single Case*. The default values should be used. As usual, progress will be shown in a DOS window. This problem took three minutes on a Xeon 3 GHz processor. When the program has finished, close the DOS window, and select *View Results/Magnetostatics*. The generation of the results view may take a few minutes.

The final step is to view the results through the menu item *View Results / Magnetostatics*. The dialog in Figure 226 controls the display. Set the controls as shown in the figure. Note that because the magnitude of the field in the iron is much higher than the field of interest, it is necessary to reduce the maximum displayed flux density. The field of interest is expected to be about 0.02 Tesla, and the maximum is set to 0.1 Tesla. Note also that clipping is turned on to allow viewing of the yz plane at x=0.



Display for:
○ Surfaces
● Parts

Vector Field Type:
● Magnetic Flux Density
○ Current Density Vector
○ Magnetic Field

Display Vector Field:
● Off ○ On

Display Vector Field Comp.:
○ Off
○ x Component
○ y Component
○ z Component
● Magnitude

Initial Mesh
Display Mesh:
● Off ○ On
☐ Show Mesh Outline

Field Parameters (T)
Calculated FluxDensity: Min -0.9157, Max 0.91087
Displayed FluxDensity: Min -0.83442, Max 0.79445
Vector Scale Factor 0.5
☑ Display Legend ● Horizontal ○ Vertical

Component Parameters (T)
Calculated FluxDensity: Min 0, Max 1.2505
Displayed FluxDensity: Min 0, Max 0.1
Number of Contours 10
☐ Show Contours
☑ Display Legend ○ Horizontal ● Vertical

Clipping State (mm)
Clipping
○ Off
● On
x Min 0 (-3500), x Max 3500 (3500)
y Min -3500 (-3500), y Max 3500 (3500)
z Min -4400 (-4400), z Max 5900 (5900)

Hide this dialog temporarily:

FIGURE 226. Magnetostatic display GUI

The field at x=0 is shown in Figure 227. The utility of the high density mesh along the beamline by the free cylinder is apparent. The irregularity of the surface outside of the free cylinder is due to the lower mesh density, and the fact that no smoothing is done in this view. The field that would be used in beam simulation is smoothed. The field is uniform across the cross section, as indicated by the uniform color.

Various other components and field quantities can be plotted by selecting the vector field type and component.

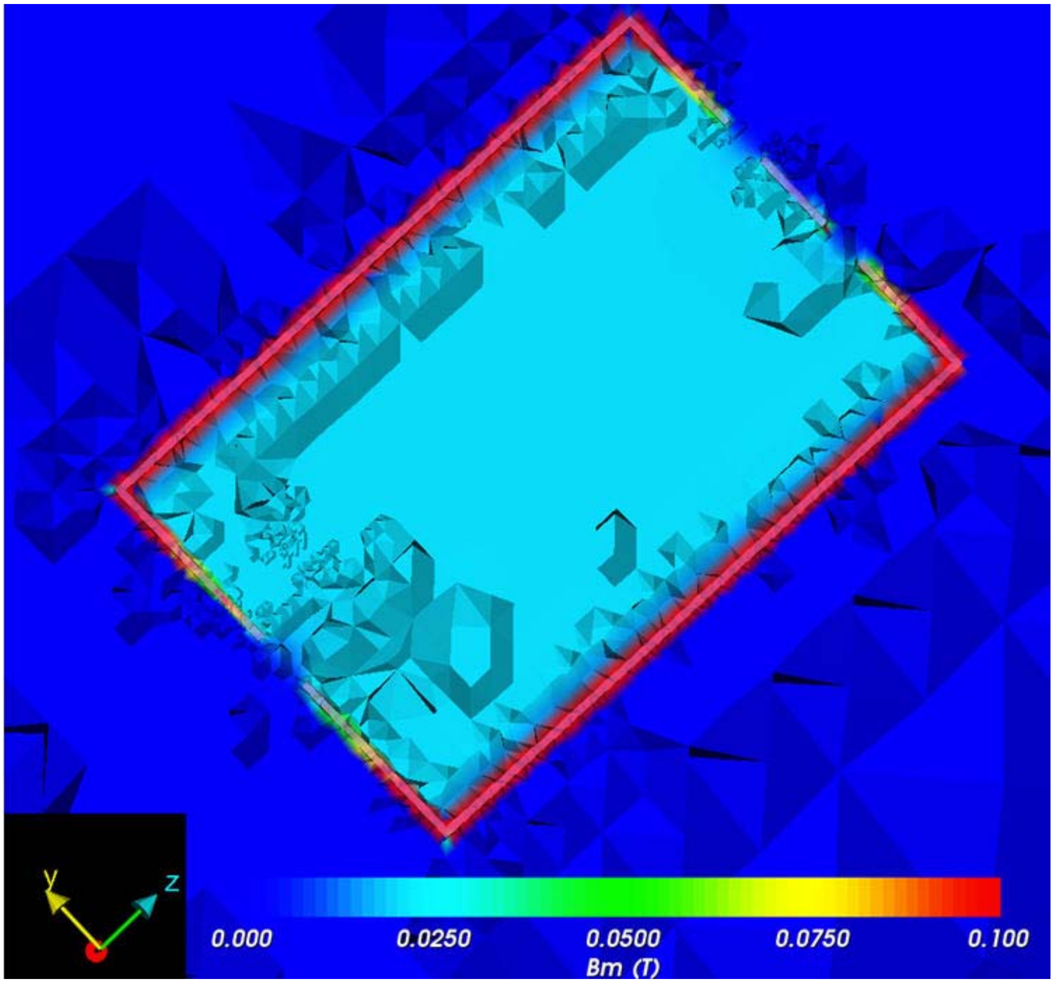
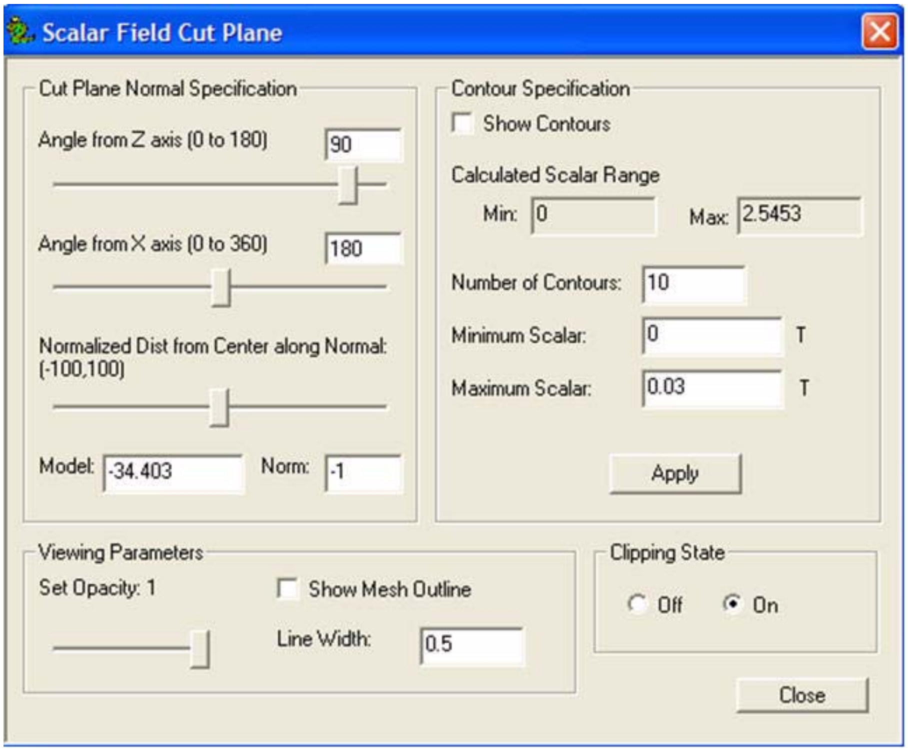


FIGURE 227. Magnetic flux density as displayed with the controls set as in Figure 226.

The field also can be viewed on a plane using ‘scalar slicing’. Turn the clipping off for this display to be most useful. The dialog controlling the scalar slicing is shown in Figure 228, and the resulting display is shown in Figure 229.


Scalar Field Cut Plane
Cut Plane Normal Specification
Contour Specification
Viewing Parameters
Clipping State
Apply
Close

Angle from Z axis (0 to 180) 90

Angle from X axis (0 to 360) 180

Normalized Dist from Center along Normal: (-100,100)

Model: -34.403 Norm: -1

☐ Show Contours

Calculated Scalar Range

Min: 0 Max: 2.5453

Number of Contours: 10

Minimum Scalar: 0 T

Maximum Scalar: 0.03 T

Set Opacity: 1

☐ Show Mesh Outline

Line Width: 0.5

○ Off ● On

FIGURE 228. Scalar cut plane GUI.

The magnetic field along a line can be plotted using *Line Plot*, which brings up the dialog shown in Figure 12. The parameters for a line at the center of the beam-line are shown in the dialog. The line plot with these parameters is shown in Figure 231. Note that the vector components of the field density as well as H and the current density J can also be plotted.

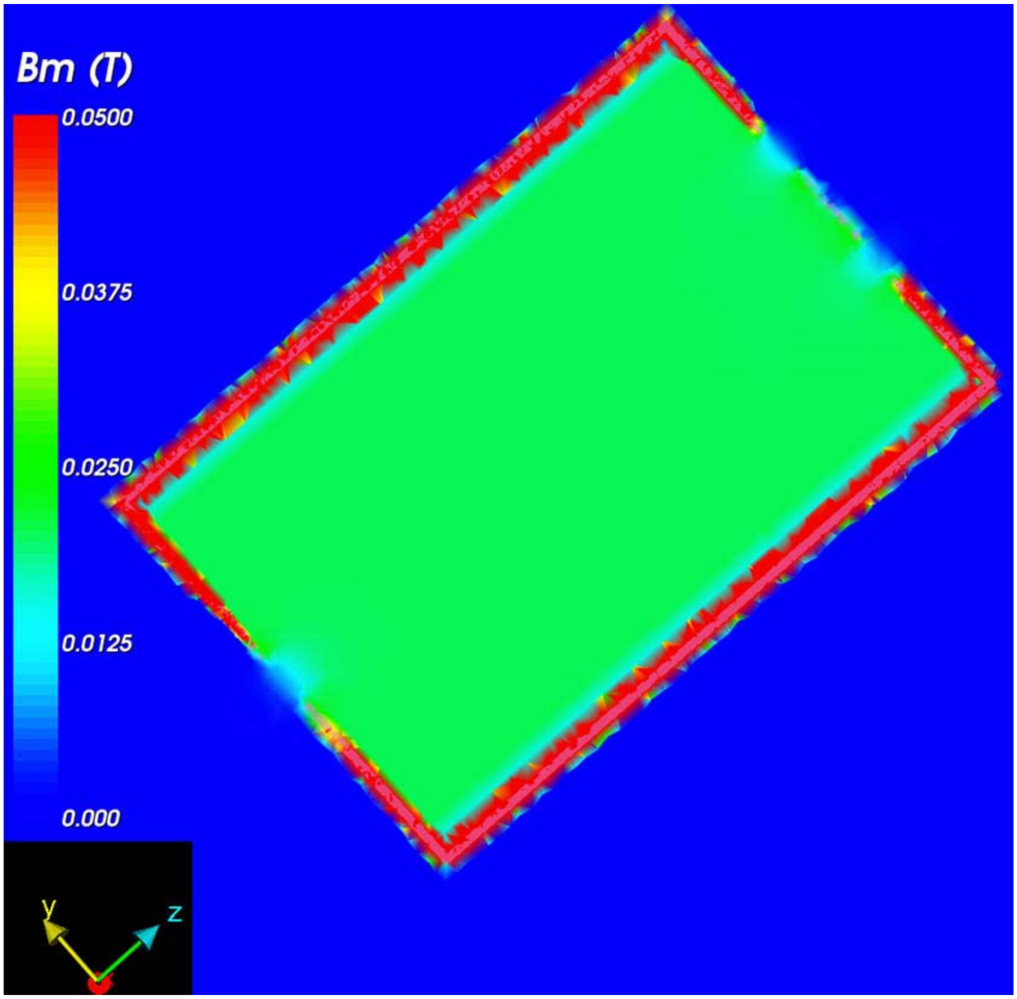
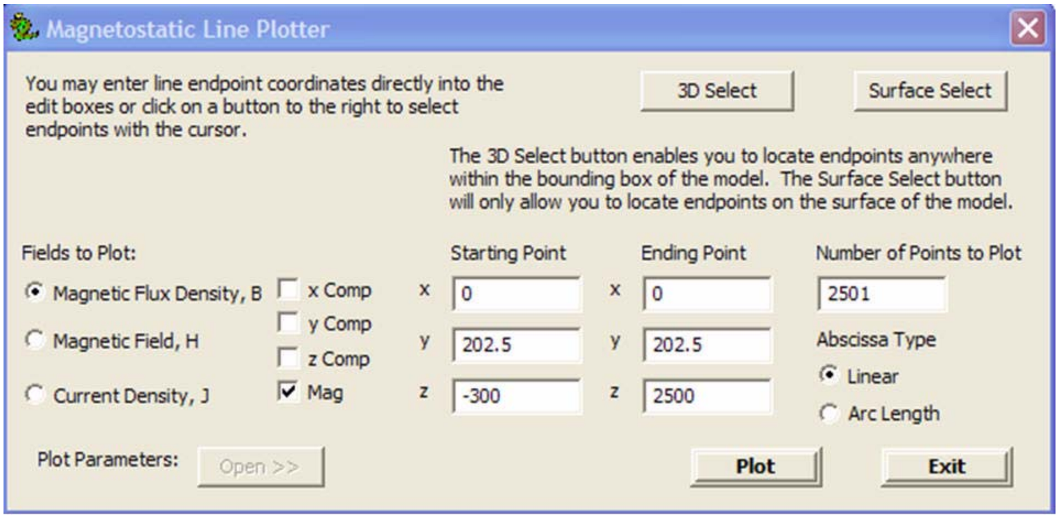


FIGURE 229. Scalar slice of the magnitude of the magnetic field density.


Magnetostatic Line Plotter
3D Select
Surface Select
Plot Parameters:
Open >>
Plot
Exit

You may enter line endpoint coordinates directly into the edit boxes or click on a button to the right to select endpoints with the cursor.

The 3D Select button enables you to locate endpoints anywhere within the bounding box of the model. The Surface Select button will only allow you to locate endpoints on the surface of the model.

Fields to Plot:
● Magnetic Flux Density, B
○ Magnetic Field, H
○ Current Density, J

☐ x Comp
☐ y Comp
☐ z Comp
☑ Mag

	Starting Point	Ending Point
x	0	0
y	202.5	202.5
z	-300	2500

Number of Points to Plot: 2501

Abscissa Type
● Linear
○ Arc Length

FIGURE 230. Line Plotter dialog

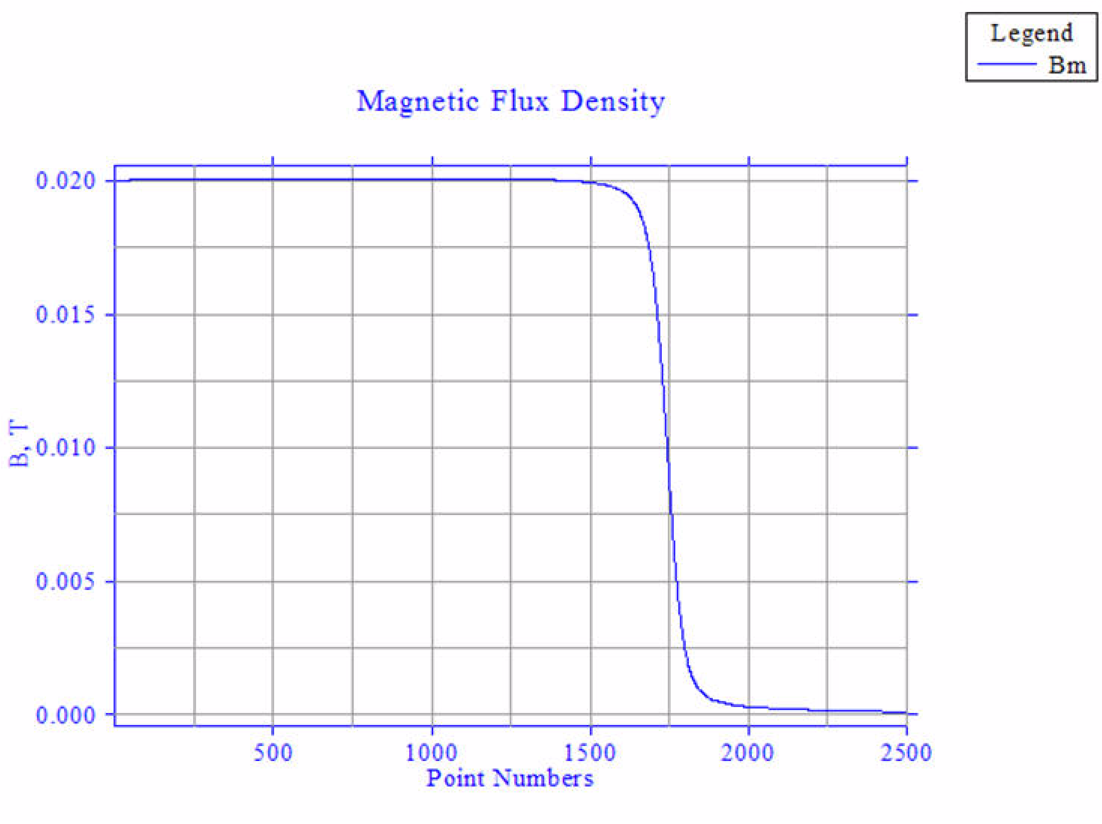


FIGURE 231. Magnetic flux density plotted along the line defined in Figure 230.

{TBD: Add additional Example cases

- *Magnetostatic*
- *Optimization case}*

5.0 Troubleshooting

To report problems or bugs, contact Thuc Bui (bui@calcreek.com).

{Is the Help-> Send Case to CCR menu item functional? If so, add a discussion here}

6.0 Appendices

This chapter contains the following Appendices

- Appendix A - Input Files
- Appendix B - Output Files (Boa-generated Files Intended for User)
- Appendix C - Internal Files (BOA-generated Files for Internal Use Only)

6.1 Appendix A - Input Files

This appendix documents the input files required by BOA, including

- Geometry File (required for all analyses)
- Injected Beam File (required in beam analyses if specifying an injected emitter)
- External Magnetic Field Files (required in Electro/Magnetostatic/Beam and Electro/Magneto/Self Field Beam analyses)
- Excel Optimization File

6.1.1 Geometry File

Each BOA analysis requires a geometry file. The geometry file is specified on the *Geometry Models Parameters* dialog (accessed via the *Model->Parameters* menu item). The file must be either an ACIS *.sat file or a ParaSolid *.x_t file. It is important to be aware of the following restrictions and conventions:

Model Units

The units used in the geometry model file dictate the units used in the BOA analysis. You do not have the option to change units from within BOA. Be sure to define the model units in the CAD package as you intend to use them in BOA. *{TBD - MAKE SURE THIS IS SAID in THE DIALOG SECTION TOO}*

Part Names

The geometry file contains parts representing the device being modeled. When BOA imports the geometry file, it uses the part names specified in that file. Part names are not editable in BOA so make sure to name parts within the CAD program as you want them to appear in BOA.

Note to beta users: The geometry file should not contain a ‘Background’ part. The newer version of BoaGui automatically creates a Background part when more than one non-Background part exists in the CAD file.

Surface Names

You are not required to name surfaces in the CAD package before exporting the model to a *.sat or a *.x_t file. Unlike with parts, you can name surfaces directly in BoaGui. However, there are distinct advantages to naming surfaces to which you plan to assign properties. These include all emission surfaces as well as surfaces that have anything to do with boundary conditions.

Often a CAD package breaks a single surface into multiple surfaces. If you do not name your surface, BOA will consider each surface of the surface as a single surface. For example, if you have a single emitting surface in the CAD package, upon exporting the CAD package might break it into, say, three surfaces. BOA will then consider it three surfaces and if you want to define emitter properties for it, you will have to do it three separate times. However, if you name the surface, BOA will be able to ‘reassemble’ the surfaces back into a single surface as you intended it and you will see the surface name in BOA. (In effect, the fact that it was broken into three pieces will be hidden from you). Therefore it is recommended that you name any surface you plan to assign properties to.

6.1.2 Injected Emitter File

BOA supports the following four file formats for the spent beam data:

- BOA file format (also referred to as CCR format),
- Magic2D file format
- Magic3D file format
- Christine file format

Boa supports multiple injected beams per analysis. In the current beta implementation, each beam from each injected plane requires a separate file for injected particles.

By convention, the extension for the Injected Beam Parameters filename is *.ibf. However, this is not a requirement.

6.1.2.1 BOA Format (or CCR Format)

There are three acceptable CCR formats the Injected Emitters:

- CCR Format 1 (the original CCR format used in early beta),
- CCR Format 2 (a newer CCR format),
- *.ppo file, a.k.a. Papa output file (One can slice the beam to obtain particle planar attribute on a plane, then use its output, the ppo file as an injected beam file.)

Format 1

The BOA file format contains the complete 3D particle attributes at the *original* injected plane. Its form is simply

pID $x[u_L]$ $y[u_L]$ $z[u_L]$ $v_x[u_L/s]$ $v_y[u_L/s]$ $v_z[u_L/s]$ I[A]

where

pID is an identifier for the particle. It must be an integer number and distinct for each particle.

x, y, z represent the initial particle position in global coordinates. The position units (shown as u_L above to indicate user length) correspond to the *Model Units* defined on the *Geometry Model Parameters* screen (e.g. if the *Model Units* are set to inches, the position coordinates are assumed to be in inches).

vx, vy, vz represent the initial particle velocity. The velocity units correspond to the *Model Units* defined in the *Geometry Model Parameters* screen (e.g. if the *Model Units* are set to inches, the velocity coordinates are assumed to be in inches/sec).

I is the particle current is in Amperes.

Example of CCR Format 1:

```
0 0.0 0.5 0.5 1.0e4 0.0 0.0 0.001
1 0.0 0.25 0.25 1.0e4 0.0 0.0 0.001
2 0.0 0.75 0.25 1.0e4 0.0 0.0 0.001
3 0.0 0.75 0.75 1.0e4 0.0 0.0 0.001
4 0.0 0.25 0.75 1.0e4 0.0 0.0 0.001
```

Format 2

A second CCR format has been added whose form is as follows:

```
# comment line
    VelocityType = explicit | normalized | relativistic
    LengthUnit = 0.0254    #unit conversion to meter, this example indicates length unit is
                           #inch
    zPlane = 0.000         #injecting plane
# if no velocity type is specified, explicit type is assumed
# if no length unit is specified, assume length unit is the same as model length unit
    q = xx     # particle charge of xx is used instead of Ip if specified
    pID x y [z] vx vy vz Ip #explicit
# or
    pID x y [z] bx by bz Ip #normalized
# or
    pID x y [z] gbx gby gbz Ip    #relativistic
```

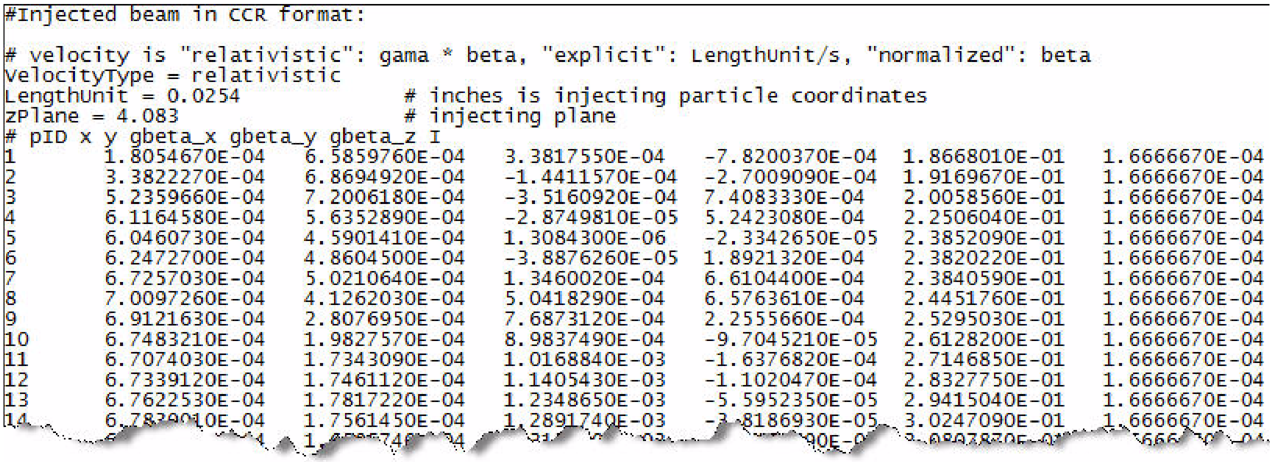
where z is not needed if zPlane is specified, pID is the particle ID,

(x,y,z) is the particle position, (vx,vy,vz) the particle velocity, (bx = vx/c, by = vy/c, bz = vz/c) normalized velocity,

(gbx = gamma bx, gby = gamma by, gbz = gamma bz) and Ip the particle current in Amp

gamma = 1/sqrt(1 - v^2/c^2)

Example of CCR Format 2:



```
#Injected beam in CCR format:

# velocity is "relativistic": gama * beta, "explicit": LengthUnit/s, "normalized": beta
VelocityType = relativistic
LengthUnit = 0.0254          # inches is injecting particle coordinates
zPlane = 4.083               # injecting plane
# pID x y gbeta_x gbeta_y gbeta_z I
1       1.8054670E-04   6.5859760E-04   3.3817550E-04   -7.8200370E-04  1.8668010E-01   1.6666670E-04
2       3.3822270E-04   6.8694920E-04   -1.4411570E-04  -2.7009090E-04  1.9169670E-01   1.6666670E-04
3       5.2359660E-04   7.2006180E-04   -3.5160920E-04  7.4083330E-04   2.0058560E-01   1.6666670E-04
4       6.1164580E-04   5.6352890E-04   -2.8749810E-05  5.2423080E-04   2.2506040E-01   1.6666670E-04
5       6.0460730E-04   4.5901410E-04   1.3084300E-06   -2.3342650E-05  2.3852090E-01   1.6666670E-04
6       6.2472700E-04   4.8604500E-04   -3.8876260E-05  1.8921320E-04   2.3820220E-01   1.6666670E-04
7       6.7257030E-04   5.0210640E-04   1.3460020E-04   6.6104400E-04   2.3840590E-01   1.6666670E-04
8       7.0097260E-04   4.1262030E-04   5.0418290E-04   6.5763610E-04   2.4451760E-01   1.6666670E-04
9       6.9121630E-04   2.8076950E-04   7.6873120E-04   2.2555660E-04   2.5295030E-01   1.6666670E-04
10      6.7483210E-04   1.9827570E-04   8.9837490E-04   -9.7045210E-05  2.6128200E-01   1.6666670E-04
11      6.7074030E-04   1.7343090E-04   1.0168840E-03   -1.6376820E-04  2.7146850E-01   1.6666670E-04
12      6.7339120E-04   1.7461120E-04   1.1405430E-03   -1.1020470E-04  2.8327750E-01   1.6666670E-04
13      6.7622530E-04   1.7817220E-04   1.2348650E-03   -5.5952350E-05  2.9415040E-01   1.6666670E-04
14      6.7839010E-04   1.7561450E-04   1.2891740E-03   -3.8186930E-05  3.0247090E-01   1.6666670E-04
```

FIGURE 232. Example of CCR format 2

Papa Output file (*.ppo file)

One can slice the beam to obtain particle planar attribute on a plane, then use its output, the ppo file, as an injected beam file of type CCR format.

{are these three options mentioned in the Injected Emitters dialog section?}

If the injected emitter requires a transformation (as specified in the boa gui) then the original injected plane will be transformed to the actual injected plane per the GUI inputs.

6.1.2.2 Magic3D Format

The Magic3D file format, like the BOA format (which is for a DC, Cartesian beam), contains the complete 3D particle attributes at the *original* injected plane. It has the form

-I[A]   x[m]   y[m]   z[m] γv_x[m/s] γv_y[m/s] γv_z[m/s]

Similar to the BOA beams, if the Magic3D injected emitter is a transformed type, its original injected plane will be transformed to the actual injected plane.

6.1.2.3 Magic2D Format

The Magic2D file format is for Polar beams. Its spent beam axis is assumed to be always the z-axis. Thus, the original injected plane will always be the rθ plane normal to the z-axis, and the direction of the beam is assumed to always point to the same direction of the positive z-axis. A Magic2D beam is always a transformed type (i.e., it always requires coordinate transformations). The format of a Magic2D beam is

-I[A] z[m] r[m] γv_z[m/s] γv_r[m/s] γv_θ[m/s]

6.1.2.4 Christine Format

The Christine spent beam can be either Cartesian or Polar beam. Like Magic2D beams, its spent beam axis is also assumed to be always the z-axis. Thus, the original injected plane will always be the xy or rθ plane normal to the z-axis, and the direction of the beam is assumed to always point to the same direction of the positive z-axis. Also like Magic2D beam, a Christine beam is always a transformed type, and always requires coordinate transformations. The format of a Polar Christine beam is

pID KE[keV] Phase[deg] r[cm] θ[deg] p_r[mc] p_θ[mc] I[A]

where pID is the particle number. KE is the particle kinetic energy, p_r and p_θ are the particle normalized momenta. The format of the Cartesian Christine beam is *assumed* to be

pID KE[keV] Phase[deg] x[xm] y[cm] p_x[mc] p_y[mc] I[A]

Note: This format still needs to be confirmed. This format is not based on an example; it is just extended from a provided example of Polar format. Thus, the Cartesian Christine file format is subject to change when the time comes to model sheet beams in Christine.

6.1.3 External Magnetic Field File

Currently BOA supports the following file types for external magnetic fields:

- Boa Magnetic Flux Density (fef file)
- CCR Imported Magnetic Flux Density (bf file)
- Maxwell 3D Imported Magnetic Flux Density
- Maxwell 2D XY Imported Magnetic Flux Density
- Maxwell 2D RZ Imported Magnetic Flux Density
- Pandira/Poisson RZ Imported Magnetic Flux Density
- 1D Axial Imported Magnetic Flux Density

This appendix section documents the 1D Axial Imported Magnetic Flux Density, Boa Magnetic Flux Density (fef file) and the CCR Imported Magnetic Flux Density (bf file) only.

6.1.3.1 1D Axial Magnetic Flux Density File Format

File Format:

```
z Min: 0.0 Max: 1.117 Number of Cells: 1117 # field separator must be a blank space in the header
0 -0.0000334038 # field separator is a blank space or a tab
0.001 -0.0000327384 # the first data field is the z coordinate
0.002 -0.0000320659 # the second data field is Bz in Tesla
...
1.117 0.0018822490 # zmax = 1.117 as in the header
```

In this example, there are 1118 data points; thus, 1117 cells or intervals. They must be equally spaced in z coordinate. Also, the header has total of nine data fields.

6.1.3.2 Boa Magnetic Flux Density (fef file) File Format

This file is generated by Boa. The file format is:

```
# xL = (x0, y0, z0) is the coordinates of the lowest corner, dx = (dx, dy, dz) is the cell size
# Nxyz = (Nx, Ny, Nz) are the number of cells per dimension
x0 y0 z0 dx dy dz Nx Ny Nz # field separator is a blank space
# Each line below contains all k-varied vector values for a fixed index
# pair (i,j) with 0 <= k <= Nz
B[ 0, 0, 0] B[ 0, 0, 1] ... B[ 0, 0,Nz] # (i=0,j=0), field separator is a blank space
B[ 0, 1, 0] B[ 0, 1, 1] ... B[ 0, 1,Nz] # (i=0,j=1)
...
B[ 0,Ny, 0] B[ 0,Ny, 1] ... B[ 0,Ny,Nz] # (i=0,j=Ny)
B[ 1, 0, 0] B[ 1, 0, 1] ... B[ 1, 0,Nz] # (i=1,j=0)
B[ 1, 1, 0] B[ 1, 1, 1] ... B[ 1, 1,Nz] # (i=1,j=1)
...
B[Nx,Ny, 0] B[Nx,Ny, 1] ... B[Nx,Ny,Nz] # (i=Nx,j=Ny)
```

6.1.3.3 CCR Imported Magnetic Flux Density (bf file) File Format

The CCR formatted file can be generated via PostPro for a Boa Magnetostatic run.

If you are using an unsupported magnetostatic solver and want to import its magnetic field into BOA, the easiest thing to do is to write a translator to convert the format to the CCR format.

6.1.4 Excel Optimization File

{TBD: get info from MR or TB}

6.2 Appendix B - Output Files

6.2.1 Planar Particle Output File (Papa output)

TBD {note that default filename can be changed; Note that papa output file can also be used as injected beam file (considered a CCR file format type}

6.2.2 Terminal Particle Output File (Enpa output)

TBD {The default name for the file is ‘n-Enpa.out’, where n corresponds to the Particle Iteration.}

6.2.3 Cibo Output File

TBD {see ibf file format}

6.2.4 Case Summary File

TBD

6.3 Appendix C - Internal Files

By convention all files post-fixed wit the character '_' are for internal usage only. Examples are CaseName_.ebm, CaseName_.cde and ModelName_E_.sat. A file pre-fixed with a number and '-' indicates this file is for that particular particle iteration number. An example is 0-CaseName_.spd, 0 stands for 0th iteration.

List of BOA-generated files:

- *.ssv files: [S]olution and [S]ource [V]olume files keep the list of the VTK readable msd files of regional (3D) electrostatic or magnetostatic fields generated from post-processing the fef files via either *Electrostatic Display Control Panel* or *Magnetostatic Display Control Panel*.
- *.sss files: [S]olution and [S]ource [S]urface files keep the list of the VTK readable mss files of surface (2D) electrostatic or magnetostatic fields generated from post-processing the fef files via either *Electrostatic Display Control Panel* or *Magnetostatic Display Control Panel*.
- *.pws files: [PoW]er [S]urface files keep the list of the VTK readable spd files of surface (2D) power density fields generated from post-processing the fef files only in Beam Optics Display Control Panel.
- *.ebm files: [E]nd [B]ea[M] files keep all terminal particle attributes automatically created after each Beam Optics Analysis.
- *.out files: Output files from various tools such as PAPA and Enpa.
- *.cde files: keep the current and potential convergences during a beam analysis. This file is used to update the Convergence plot.

- *.tdb files: [T]empo [D]ata [B]ase files to keep track of the fef and jc files available after each iteration during the analysis. These fef and jc files are listed only for those iterations that users select to store (outputable iteration). The 0th, 1st and last iteration files are always stored. Noted for either electrostatic or magnetostatic (no beam) analysis, there is only 0th iteration.
- *trj.tdb files: keep track of trajectory and power density fef files available after each ouputable iteration during a beam analysis.
- *.fef files: [F]inite [E]lement [F]ield files keep the solution fields produced by either magnetostatic or electrostatic field solver. *(Note: The clean up feature removes all *.fef files generated by the case, but does not remove those imported into the case, such as those generated by another case)*
- *.jc files: Cathode current density files: only exists in a beam analysis. It stores the current density on emitter surfaces.
- *.trj files: Particle trajectories produced by a beam analysis.
- *.msd files: 3D volumetric VTK files keeping the 3D fields created from fef files for display.
- *.mss files: 2D surface VTK files keeping the 2D fields created from fef files for display.
- *.spd files: 2D surface VTK files keeping the 2D power density field calculated from the power density fef file, which actually derived from the trj file.
- *.res: file from the conjugate gradient solver
- *.sle stores user's selected entities during post processing